DOCUMENT RESUME

ED 391 901 CE 070 758

TITLE	Basic CNC Operation. Training Workbook [and] Assessment and Training Guide [and] Hands-on Assessment.
INSTITUTION	Anoka-Hennepin Technical Coll., Minneapolis, MN.
PUB DATE	94
NOTE	401p.; For related documents, see CE 070 760, CE 070 763, and CE 070 765-768.
PUB TYPE	Guides - Classroom Use - Teaching Guides (For Teacher) (052)
EDRS PRICE	MF01/PC17 Plus Postage.
DESCRIPTORS	Adult Education; *Adult Vocational Education; Behavioral Objectives; *Competency Based Education; *Computer Assisted Manufacturing; *Equipment Utilization; Learning Activities; *Machine Tool Operators; *Numerical Control; Programming; Troubleshooting; Workbooks

ABSTRACT

This workbook is intended for students taking a course in basic computer numerical control (CNC) operation that was developed during a project to retrain defense industry workers at risk of job loss or dislocation because of conversion of the defense industry. The workbook contains daily training guides for each of the course's 13 sessions. Among the topics covered in the course sessions are the following: CNC machine terms, machine specifications, the CNC coordinate grid system, and absolute/incremental programming; toolholder manufacturing processes, toolholding systems, and toolholder styles; functions and keys on CNC operator panels and writing/editing CNC programs; alarm codes and messages; program codes and preset tool methodology; operating CNC machines; using programming codes and locating errors in programs; operating a horizontal machine center; defining and determining machinability factors (speed, feed, and depth of cut); operating a CNC lathe; and troubleshooting CNC machine tools. Each daily training guide includes some/all of the following: session objectives, diagrams and specifications of the machine(s) introduced during the session, information sheets, and learning activities. An assessment and training guide and hands-on assessment for students in the course are also provided. (MN)

****
* Reproductions supplied by EDRS are the best that can be made *
* from the original document. *
****

ED 391 901

Basic CNC Operation

Training Workbook

U.S. DEPARTMENT OF EDUCATION
Office of Educational Research and Improvement
EDUCATIONAL RESOURCES INFORMATION CENTER (ERIC)

☑ This document has been reproduced as received from the person or organization originating it.

☐ Minor changes have been made to improve reproduction quality.

- Points of view or opinions stated in this document do not necessarily represent official OERI position or policy.

"PERMISSION TO REPRODUCE THIS MATERIAL HAS BEEN GRANTED BY

I Strand

TO THE EDUCATIONAL RESOURCES INFORMATION CENTER (ERIC)."

CE070 758

BEST COPY AVAILABLE

Section One

Your Notes:

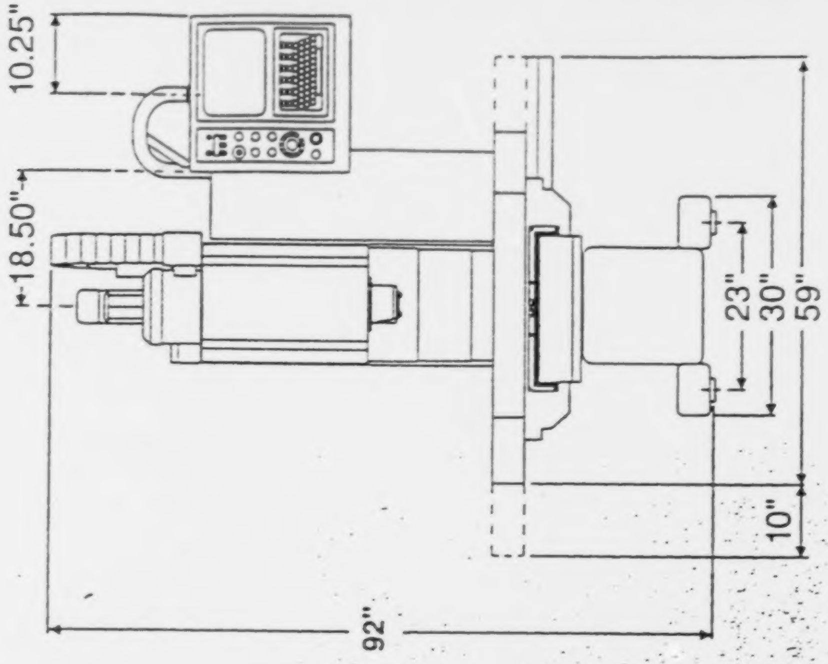
Training Daily Guide

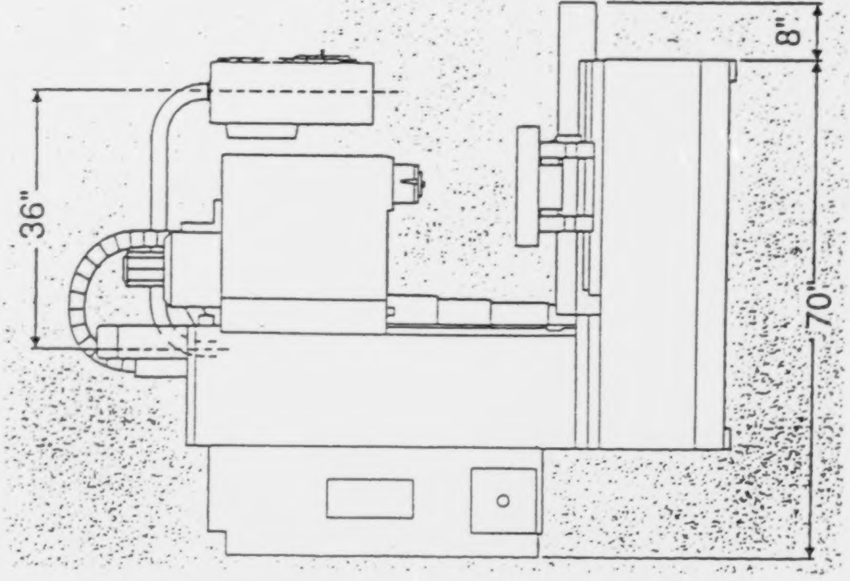
Session One

During this two hour session, participants will:

1. Locate, identify and explain the purpose of the basic components of CNC machine terms.
2. Explain machine specifications.
3. Explain machine home and the coordinate grid system and how it relates to the CNC machine.
4. Explain the four most common axes: X, Y, Z, and B.
5. Define absolute /incremental programming and the use of coordinates and signed numbers.

CNC Machine


10.25"
18.50"
23"
30"
59"
10"
92"



Machine Specification

Axes Travel

Axes Travel ______________________________

Table (X axis) ______________________________

Saddle (Y axis) ______________________________

Head (Z axis) ______________________________

Additional 'Z' stroke for toolchange ______________________________

Positioning Speed ______________________________

Auto (X & Y) ______________________________

AUTO (Z) ______________________________

Manual (X, Y, & Z) ______________________________

Feedrate Range ______________________________

Minimum increment ______________________________

Machine Specification

Table

Table____

Working Surface__________________________________

T Slots ______________________________________

T Slots Size ___________________________________

Height above Floor________________________________

Maximum Table Load______________________________

Spindle

Spindle Drive ___________________________________

Spindle Speed Range _____________________________

Spindle Speed Control_____________________________

Spindle Diameter ________________________________

Spindle Taper___________________________________

Machine Specification

Spindle

Tool Holder Clamping ______________________________

____

Tool Holder ______________________________

____

Spindle Speed Range ______________________________

____

Minimum Distance Spindle Nose to Table Top ______________________________

____

Maximum Distance Spindle Nose to Table Top ______________________________

____

Automatic Tool Changer

Magazine Capacity ______________________________

____

Tool Selection ______________________________

____

Maximum Tool Diameter ______________________________

____

Maximum Tool Weight ______________________________

____

Maximum Total Weight of all Tools ______________________________

____

Maximum Tool Length ______________________________

____

Machine Specification

Automatic Tool Changer

Tool Change Time ______________________________

____

Tool Change Time Chip to Chip ______________________________

____

Standard Equipment

Spindle Orientation______________________________

____

Flood Coolant______________________________

____

Automatic Centralized Lubrication System ______________________________

____

Full Machine Guarding______________________________

____

Slideway Protection ______________________________

____

Swarf Collection Trays ______________________________

____

Low Voltage Worklight ______________________________

____

Machine Specification

Accuracy

Positioning Accuracy ____________________

____

Repeatability ____________________

____

Mains Services Requirements

Electrical Supply____________________

____

Standard Electrical Supply ____________________

____

Low Volt Option Electrical Supply ____________________

____

Compressed Air ____________________

____

Machine Specification

STANDARD SPECIFICATION

a) System Features

- 4 axis control with automatic acceleration and deceleration.
  - 3 axis simultaneous linear interpolation (XYZ)
  - Multi Quadrant circular interpolation (XY, YZ, XZ)
- Input range
  - 0.001 to 9999.99mm (XYZ)
  - or 0.0001 to 999.9999 inch (XYZ)
- Part programme storage length 120m
- Registered programs 63
- Tool offset memory 99
- ISO/EIA Automatic recognition
- 9" CRT Mono chrome screen with MDI keyboard
- Feedrate command Direct programming of mm/min or inch/min
- Feedrate override 0 to 150%
- Rapid traverse override 0, 25, 50, 100%
- Spindle speed override 50% to 120% in 10% steps
- Override cancel
- Automatic co-ordinate system setting
- Absolute/incremental command
- Program number display/search 4 digits
- Sequence number display/search 4 digits
- Main program/subprogram (Subprogram: 2 levels)
- Optional block skip
- Stored pitch error compensation
- Backlash compensation
- Servo off
- Cycle start/feed hold
- Buffer register
- Reset
- Manual pulse generator multiplier X1, X10, X100
- Machine lock (all axes)
- Dry run
- Single block
- Part program storage and editing
- Data protect switch
- Self diagnosic function
- Emergency stop
- Stored stroke check 1
- Interlock error display
- Status output: CNC ready, servo ready, alarm, distribution end, automatic operation
- Automatic operation start-lamp, feed hold
- Digital Servo Control

b) Programming Features

- Pocket calculator type decimal point input
- Background edit
- Inch/metric operation (G20/G21)
- Exact stop/dwell G04/G09/G61
- Reference point return Manual automatic (G27, G28, G29)
- Second reference point return G30
- Co-ordinate system setting G92
- Feedrate F5 digit direct command
- Spindle speed S4/S5 digit command

Machine Specification

Tool selection T4 digits
Miscellaneous function M3 digits
Skip function G31
Tool length compensation G43, G44, G49
Programme stop/program end M00/M01/M02/M30
Canned cycles for drilling boring G73, G74, G76, G80 to G89
Cutter compensation C (G40-G59)
Mirror image (M21 to M24)
Work co-ordinate system (G54-G59)
Programme input of offset date (G10)
Absolute/incremental (G90/G91)

Scoping Options

TSUDAKOMA MACHINING CENTER ATTACHMENTS

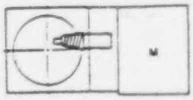
		Model	テーブルサイズ（φmm）
NC Rotary Tables	1	RNCS	225
	2	RNCX	125 ～ 320
	3	RNC	800～2000
	4	RNC.5	320 ～ 500
	5	RNCV	160～1500
	6	RNCM	250 ～ 630
	7	RNCB	250～1250
	8	RNCK	320～1250
	9	RNCBK	250～1250
NC Tilting Rotary Tables	10	TRNC.B	160～ 320
	11	TTNC	450～1000
	12	THNC	250～ 320
NC Dividing Heads	13	RDH	With controller
Automatic Dividing Heads	14	RD	Air driven
Multi-Spindle		RNCV-N	160 ～ 400
		DNC-N	135 ～ 200
		TTNC-N	200 ～ 450
Numerical Control (For NC Rotary Tables)		TPC2	
		TPC-Jr	
Auto Connector System		Auto connector	
Others		Built-In and Special Rotary Tables	

(Many deliveries have been made. Consult a Tsudakoma sales person.)

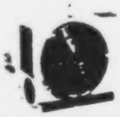
M Motor

1 RNCS

- Super High Speed, 500r.p.m.
- NC Rotary Table with High speed continuous rotation and various indexing function

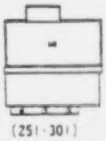


2 RNCX

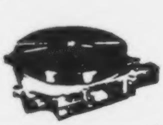


- Standard type only
- Low price
- Rear motor mounted type

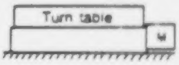
(251 · 301)



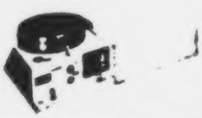
3 RNC



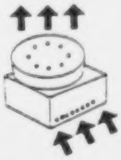
- Horizontal positioning only
- Machines large and heavy workpieces


Turn table
M

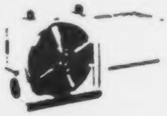
4 RNC□□5 SERIES



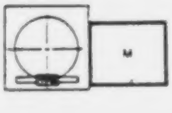
- With 8 ports rotary joint
- Excellent water resistance
- Maintenance free



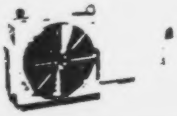
5 RNCV



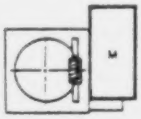
- Standard type
- Easy application for multi-rotary tables



6 RNCM



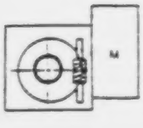
- Compact design without splash guard interference



7 RNCB

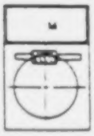
- Table center has a large thru-hole
- Machines long workpieces

Applications: Perfect for installing of an automatic power chuck, pull stud device, rotary joint device and others



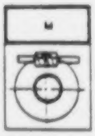
8 RNCK

- For horizontal machining centers



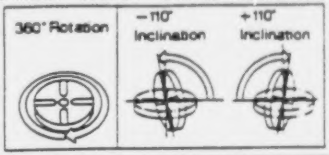
9 RNCBK

- For horizontal machining centers
- Table center has a large thru-hole



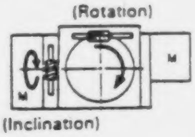
10 TRNC

- Unmovable motor location
- Wide inclination


360° Rotation
−110° Inclination
+110° Inclination

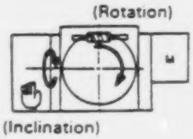
11 TTNC

- Rotation and tilting are both numerically controlled
- Complicated polyhedron machinings are possible


(Rotation)
M
M
(Inclination)

12 THNC

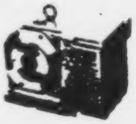
- NC tilting rotary table with manual tilt
- Low price / high quality


(Rotation)
M
(Inclination)

13 RDH

- Compact
- Heavy-duty clamping
- Large allowable workpiece inertia
- High precision
- Easy cable connection

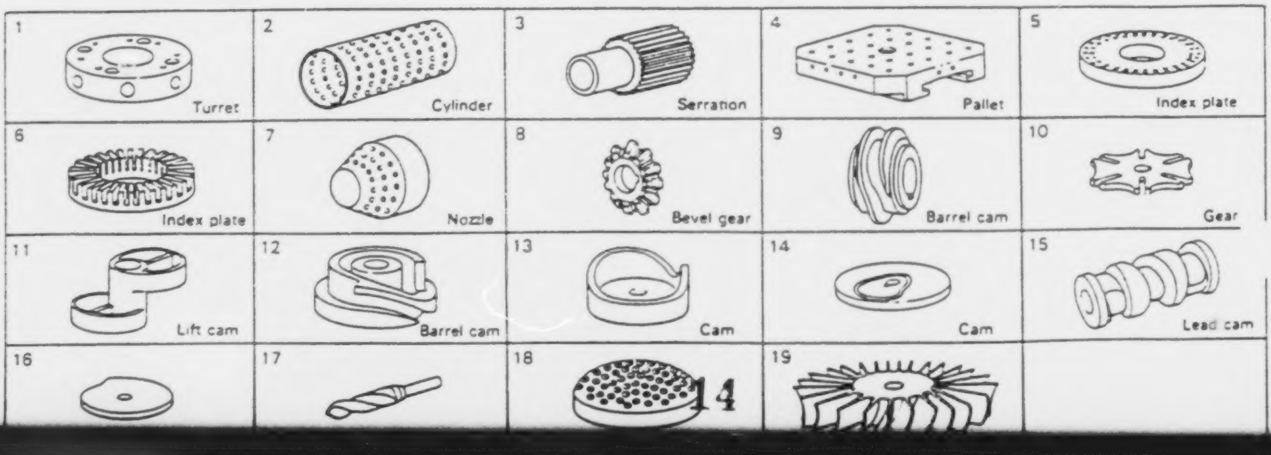
14 RD



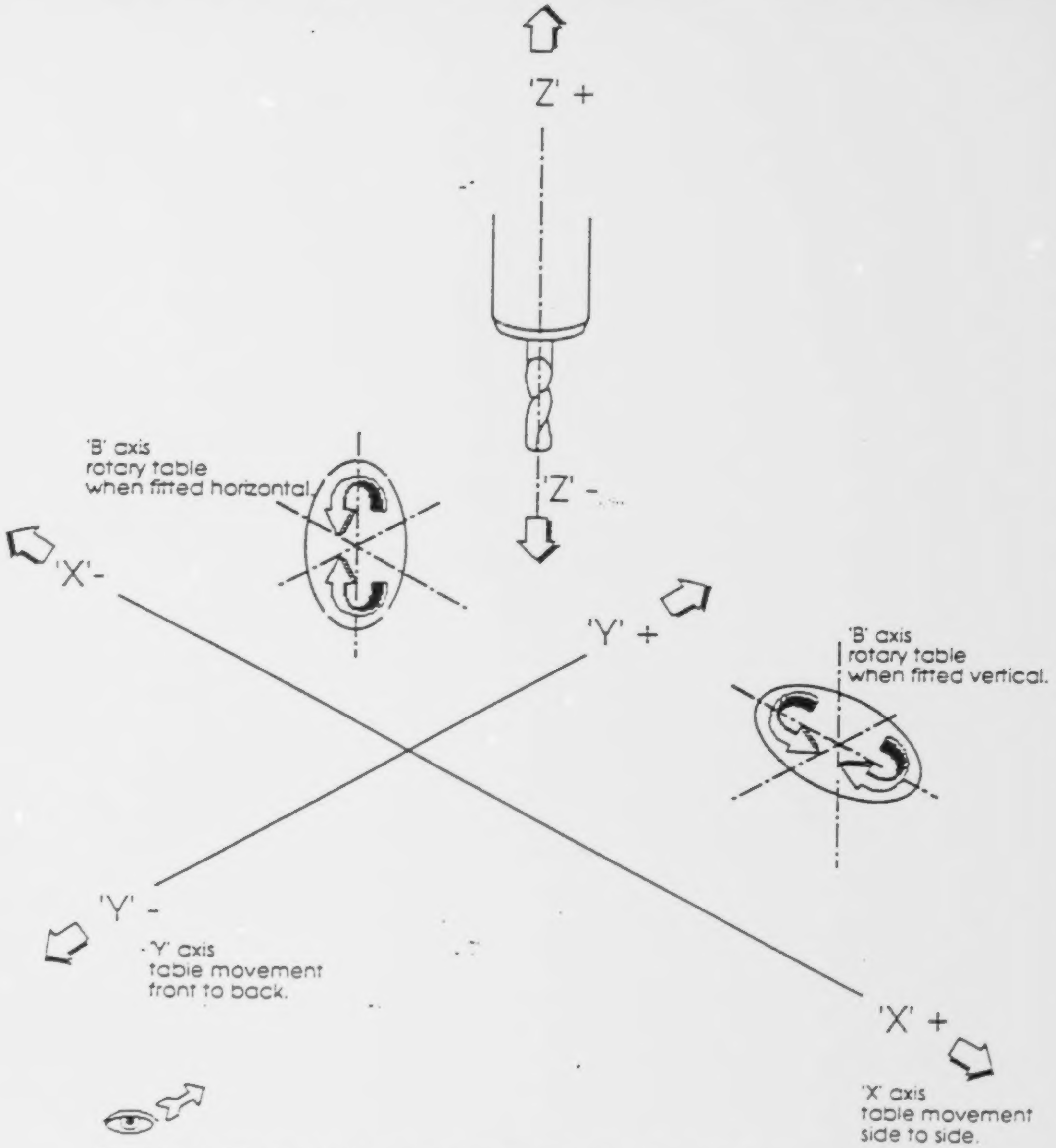
- Low price
- Quick delivery
- High speed
- High accuracy cross roller bearing and coupling attached

Workpiece Examples

BEST COPY AVAILABLE


1 Turret
2 Cylinder
3 Serration
4 Pallet
5 Index plate
6 Index plate
7 Nozzle
8 Bevel gear
9 Barrel cam
10 Gear
11 Lift cam
12 Barrel cam
13 Cam
14 Cam
15 Lead cam
16
17
18
19

Tool Motion

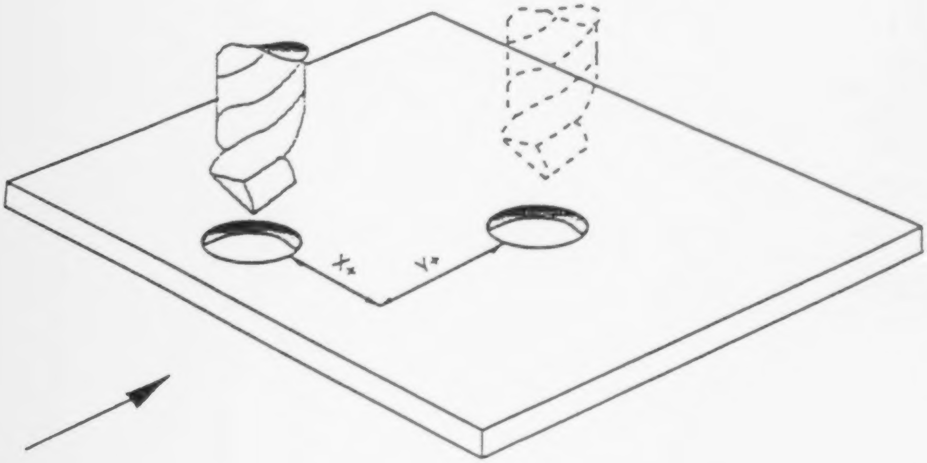

'Z' +
'B' axis
rotary table
when fitted horizontal.
'Z' -
'X'-
'Y' +
'B' axis
rotary table
when fitted vertical.
'Y' -
'Y' axis
table movement
front to back.
'X' +
'X' axis
table movement
side to side.

The nomenclature shown assumes viewing from the Operators Control Panel.

The Bridgeport machine has three axes that are under numerical control.
Each axes can move independently, the direction being designated as a plus or minus movement.

Tool Motion

When we programme a numerical controlled machine, we always consider that the cutting tool is moving.

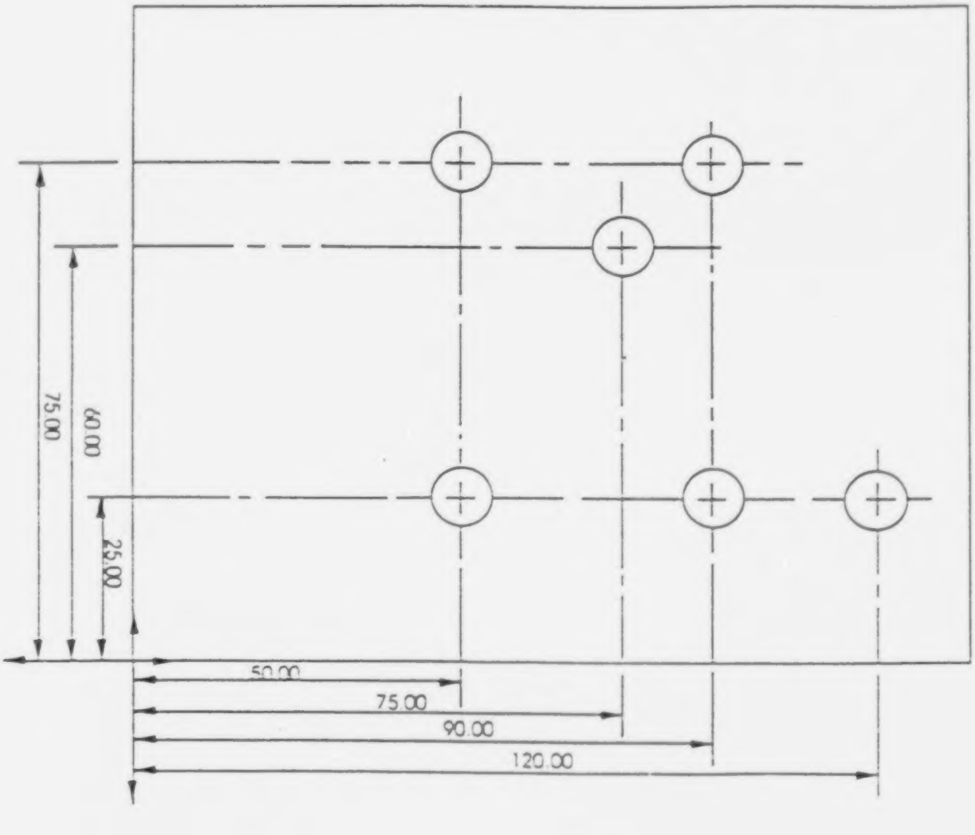


FRONT OF MACHINE

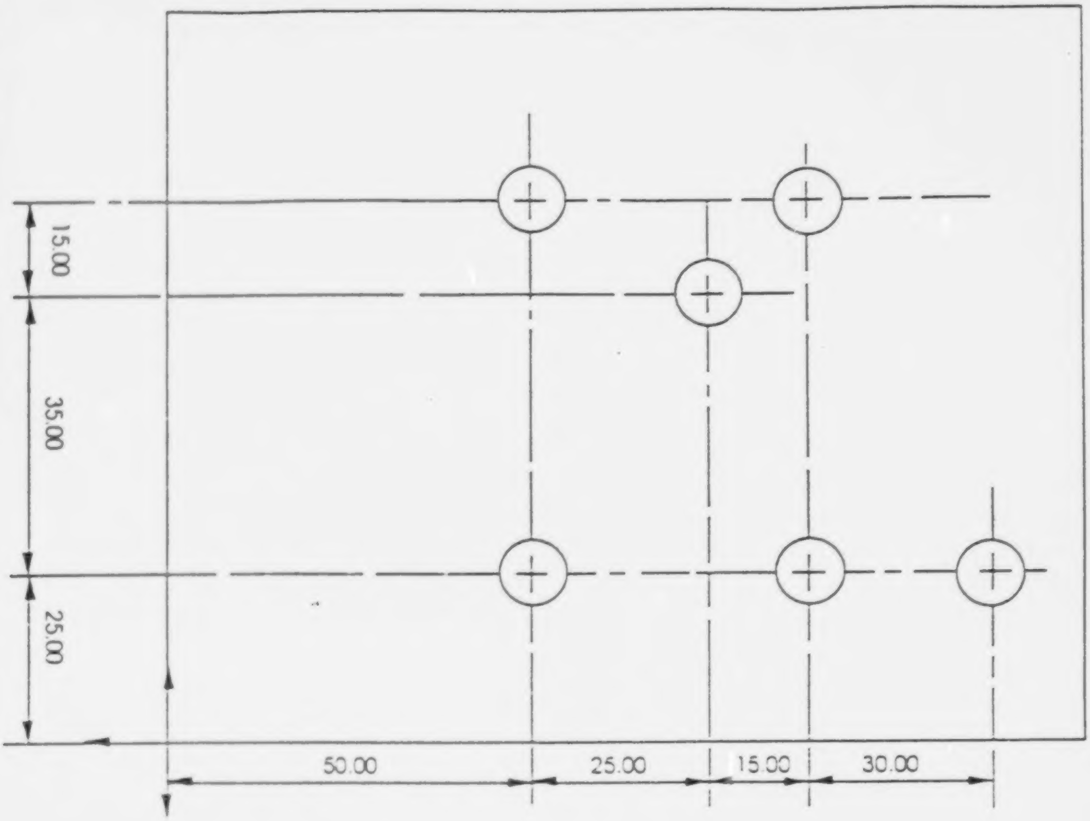
Absolute (G90) Coordinate Programming

In Absolute (G90) all dimensioning is taken from a fixed point which is specified by the programmer (see Figure 5).

Figure 5 Absolute Co-ordinate Programming


75.00
60.00
25.00
50.00
75.00
90.00
120.00

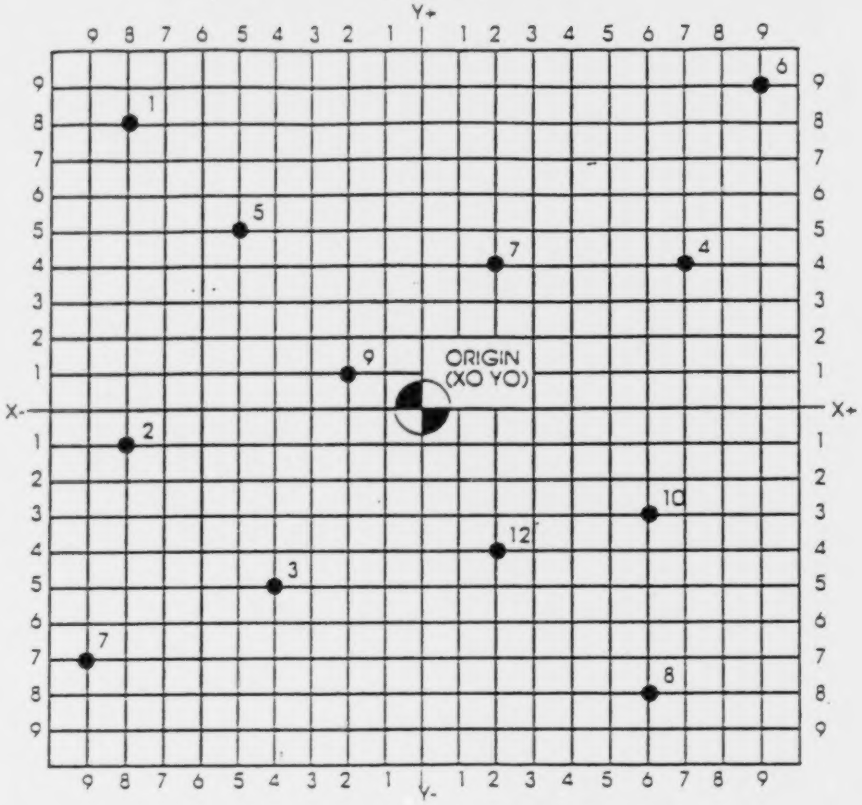
Incremental (G91) Co-ordinate Programming



15.00
35.00
25.00
50.00
25.00
15.00
30.00

Absolute and Incremental Programming

EXAMPLE 1


Y+
Y-
X-
X+
9 8 7 6 5 4 3 2 1 1 2 3 4 5 6 7 8 9
9 8 7 6 5 4 3 2 1 1 2 3 4 5 6 7 8 9
ORIGIN
(XO YO)
1
2
3
4
5
6
7
7
8
9
10
12

ABSOLUTE

6 to 1 X__________ Y __________
5 to 2 X __________ Y __________
8 to 3 X __________ Y __________
2 to 4 X __________ Y __________
4 to 5 X __________ Y __________
5 to 6 X __________ Y __________
12 to 7 X __________ Y __________
10 to 8 X __________ Y __________
4 to 9 X __________ Y __________

INCREMENTAL

6 to 1 X__________ Y __________
5 to 2 X __________ Y __________
8 to 3 X __________ Y __________
2 to 4 X __________ Y __________
4 to 5 X __________ Y __________
5 to 6 X __________ Y __________
12 to 7 X __________ Y __________
10 to 8 X __________ Y __________
4 to 9 X __________ Y __________

EXERCISE: Find the absolute and incremental co-ordinates of the points listed above.

ABSOLUTE SYSTEM: A numerical control system in which all co-ordinate locations are programmed from a fixed or absolute zero point (origin).

INCREMENTAL: A numerical control system in which each co-ordinate location is given in terms of distance and direction along rectangular axes from the previous

Linear Interpolation(G00/G01)

The axes of the machine will move in linear at either Rapid or Feedrate traverse rates.

Any movement preceded by:

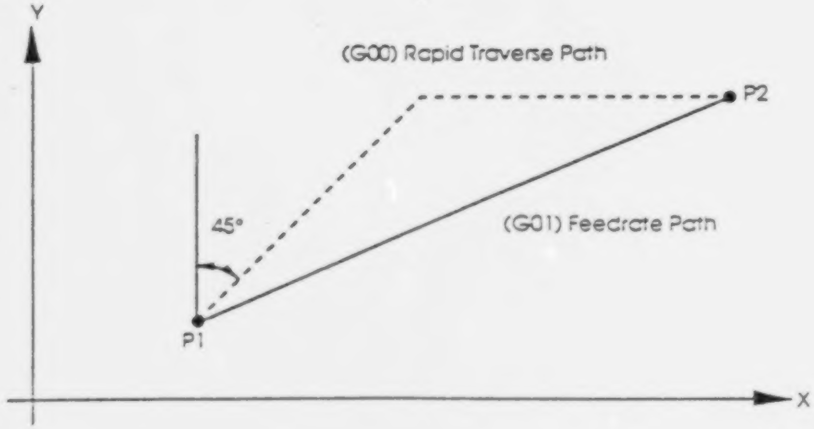
a) G00 will occur at RAPID TRAVERSE

b) G01 will occur at FEEDRATE

NOTE: These commands are MODAL and will stay in effect until changed.

Maximum of 3 axes can be programmed in one BLOCK.

Figure 9 Linear Interpolation


Y
(G00) Rapid Traverse Path
P2
45°
(G01) Feedrate Path
P1
X

Circular Interpolation (G02/G03)

Circular interpolation can be performed in any of three planes (G17, G18, G19).

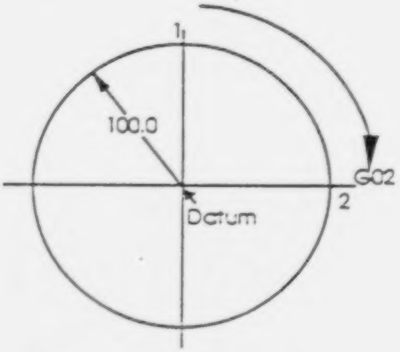
There are two directions in which arc are produced - G02 Clockwise and G03 Counter Clockwise.

The information required to move in an arc involves four 'word' addresses. Assuming G17 plane is used, the program would be N14 G02 X100.0 Y0.0 10.0 J-100.0

i.e.
G02 - Clockwise movement
X100.0 - Tool finishing position in X
Y0.0 - Tool finishing position in Y
10.0 - Arc offset in X axis
J-100.0 - Arc offset in Y axis

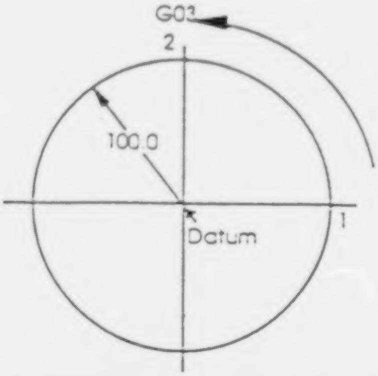
Figure 10 Circular Interpolation

Clockwise Direction



Again using G17, the program for counter clockwise movement would be:
N15 G03 X0.0 Y100.0 I-100.0 J0.0

Counter Clockwise Direction



Plane Selection (G17, G18, G19)

The Interact can perform circular interpolation in 3 planes. Either:

a) X and Y (G17) see figure 11

b) X and Z (G18) see figure 12

c) Y and Z (G19) see figure 13

NOTE: Plane selection is MODAL and once selected will stay in effect until another plane is selected.

Figure 11 G17 XY Plane (Plan view above spindle)

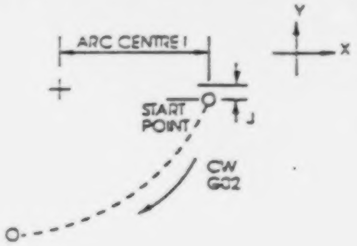

Y
X
ARC CENTRE I
J
START POINT
CW
G02

Figure 12 G18 ZX Plane (View from behind spindle)

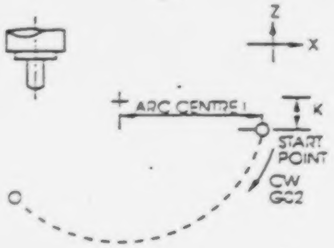
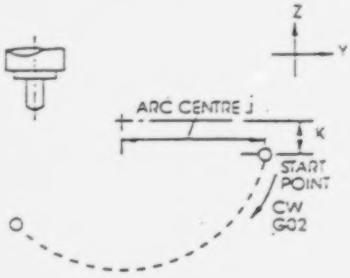

Z
X
ARC CENTRE I
K
START POINT
CW
G02

Figure 13 G19 YZ Plane (View from table end)


Z
Y
ARC CENTRE J
K
START POINT
CW
G02

Arc Offsets

Arc offsets are addressed using the 'Words'.

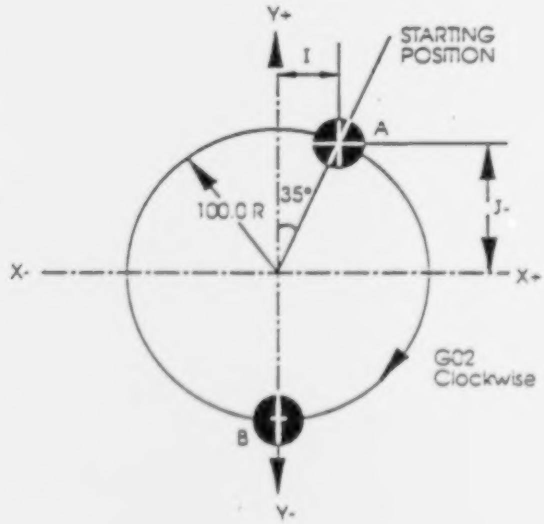
I - is the sign distance parallel along the X axis from the starting position of the arc to the arc centre.

J - is the sign distance parallel along the Y axis from the starting position of the art to the arc centre.

K - is the sign distance parallel along the Z axis from the starting position of the arc to the arc centre.

The 'word' values describe the direction in which the centre of arc lies in relation to the starting position. Values are automatically assumed to be +(plus). If a -(minus) value is required it MUST be designated. See Figure 14.

Figure 14 Arc Offsets



Explanation:

In the X axis, the circle is in the X negative direction.
In the Y axis, the circle is in the Y negative direction.

Example:

Block of program required to move from A to B:

```
N15 G02 X0.0 Y-100.0 I-57.357 J-81.915
```

Radius Command (R)

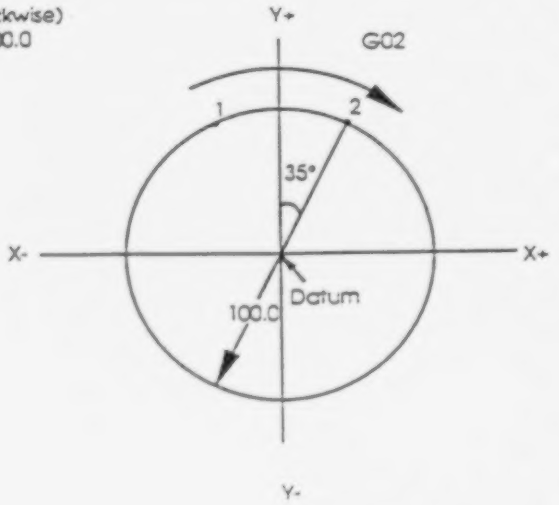
This command is used as an alternative to replace the I and J words used in circular interpolation.

Example: N15 G02 X0.0 Y-100.0 R100.0

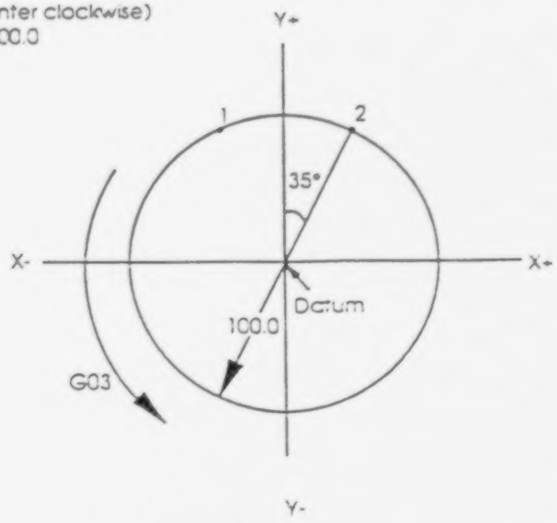
NOTE: The +R word can only be used for arcs up to and include 180°.
The -R word can only be used for arcs greater than 180°, and less than 360°.

Figure 15 Radius Command

To move from Position 1 to Position 2 (clockwise)
N100 G02 X57.357 Y81.915 R100.0



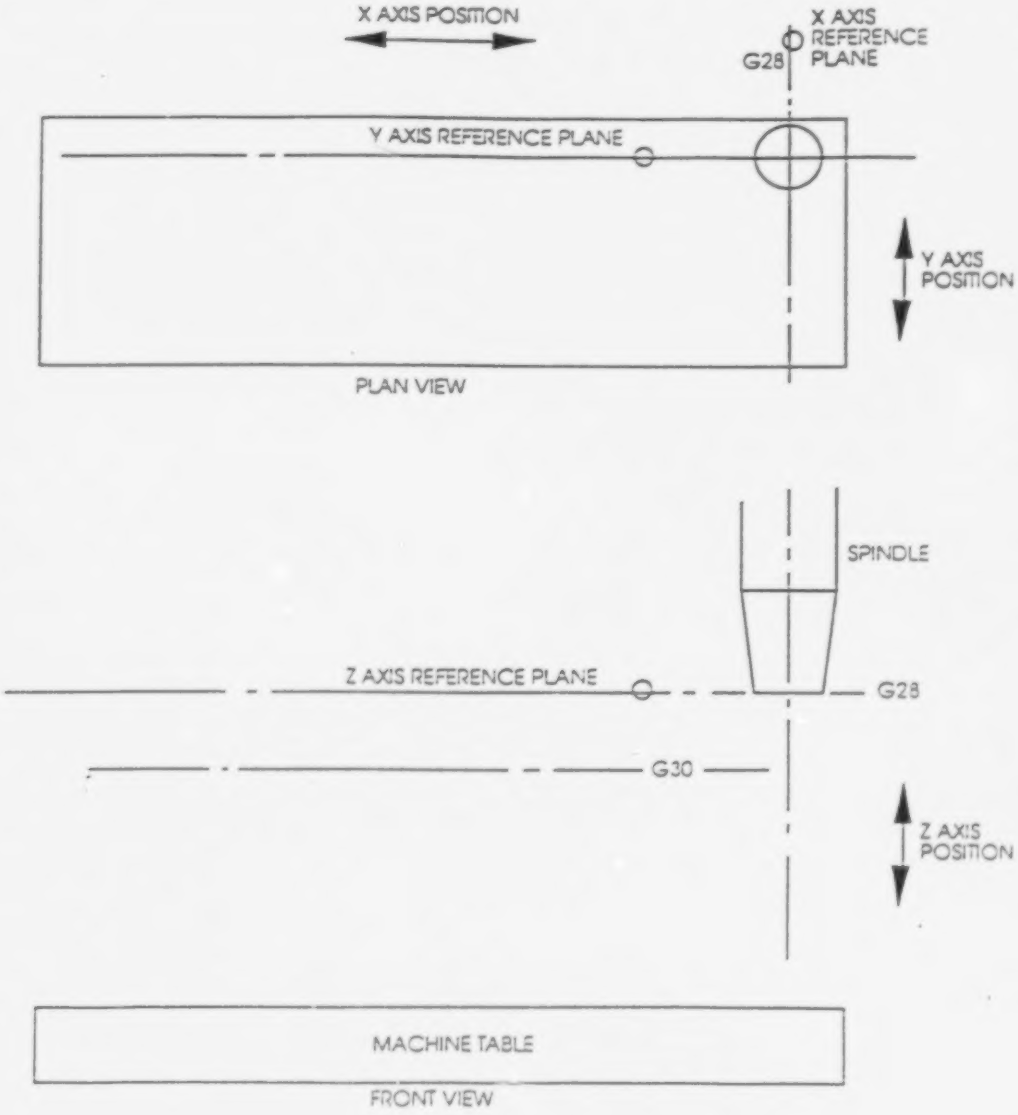
To move from Position 1 to Position 2 (counter clockwise)
N105 G03 X57.357 Y81.915 R-100.0



Machine References (G28/G30)

The machine has its own "Machine Reference co-ordinate System", from which the maximum travels of the X, Y and Z axes are measured.

Figure 16. Spindle positions at Machine Reference in X, Y and Z axes.


X AXIS POSITION
X AXIS REFERENCE PLANE
G28
Y AXIS REFERENCE PLANE
Y AXIS POSITION
PLAN VIEW
SPINDLE
Z AXIS REFERENCE PLANE
G28
G30
Z AXIS POSITION
MACHINE TABLE
FRONT VIEW

Z Axis G28 & G30 Reference Planes

There are 2 reference planes in the Z axis, to which the spindle will return automatically:

1. G28 plane which is at the Machine reference location.
2. G30 plane is where all toolchanges occur.

Program Format for G28 and G30 reference return planes.

The G28 command constitutes 2 movements:

a) Move to dimension

b) Move to reference

example N50 G28 G91 Z0

The first movement to take place is an incremental move of Z0.

The second movement will return directly to the reference plane.

Care should be taken when programming this move in the absolute mode, as the Z dimension must be clear of the workpiece.

N50 G28 G90 Z150.0

The G30 command works in the same way as the G28, but is only to be used for toolchange purposes.

i.e. N60 G30 G91 Z0 T4 M6

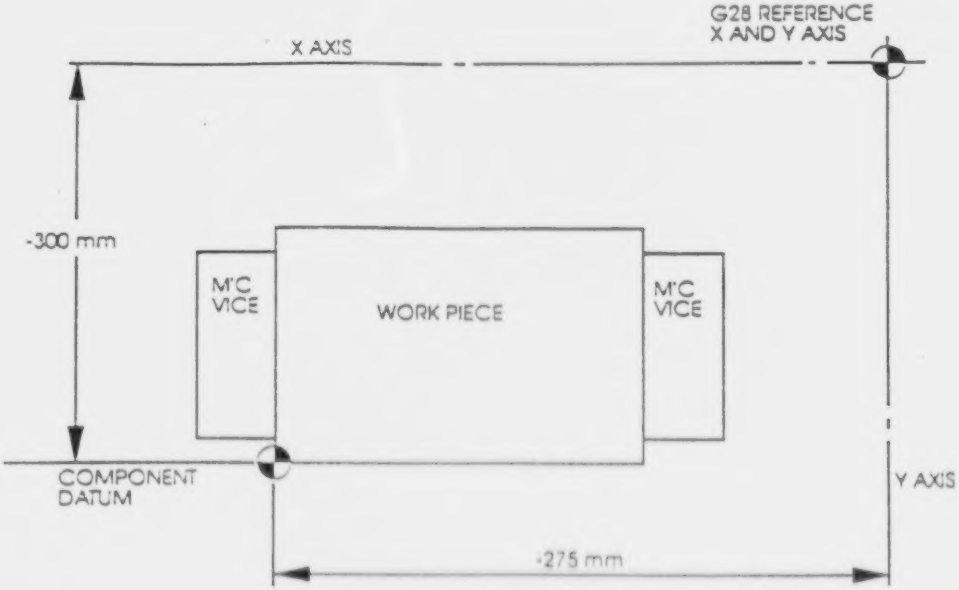
After movement to the G30 reference plane, the machine will initiate a toolchange cycle, selecting tool number 4.

Work Co-ordinate System Programming (G54-G59)

The work co-ordinate system allows for the setting of datums relative to the machine reference coordicate system.

X and Y axis values for use with G54-G59 co-ordinate system.

Figure 17


G28 REFERENCE
X AND Y AXIS
X AXIS
-300 mm
M'C VICE
WORK PIECE
M'C VICE
COMPONENT DATUM
Y AXIS
-275 mm

When the position of the component datum has been determined, it can be entered into the relative work offset register.

This can be done in 2 difference ways:

a) Manually, whereby the dimensions can be entered directly through the keyboard.

b) Programmed, whereby the dimensiosn can be entered into the program in the following format;

N10 G10 G90 L2 PO1 X-275.0 Y-300.0

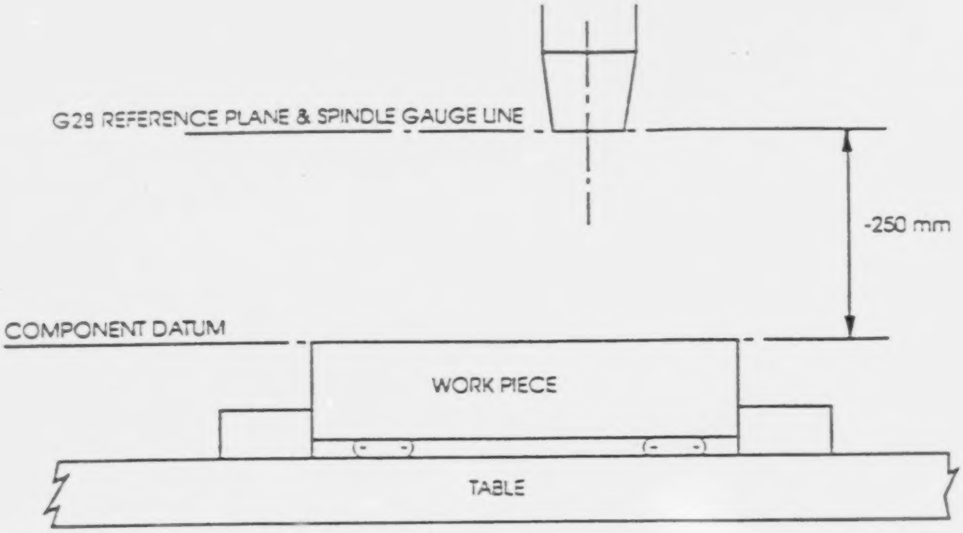
Explanation:

G10 L2 = Work co-ordinate offset input
PO1 = Specifies offset register G54
PO2 = G55 - G59

Z Axis Values

Z AXIS VALUES FOR USE WITH G54-G59 COORDINATE SYSTEMS

Figure 18


G28 REFERENCE PLANE & SPINDLE GAUGE LINE
-250 mm
COMPONENT DATUM
WORK PIECE
TABLE

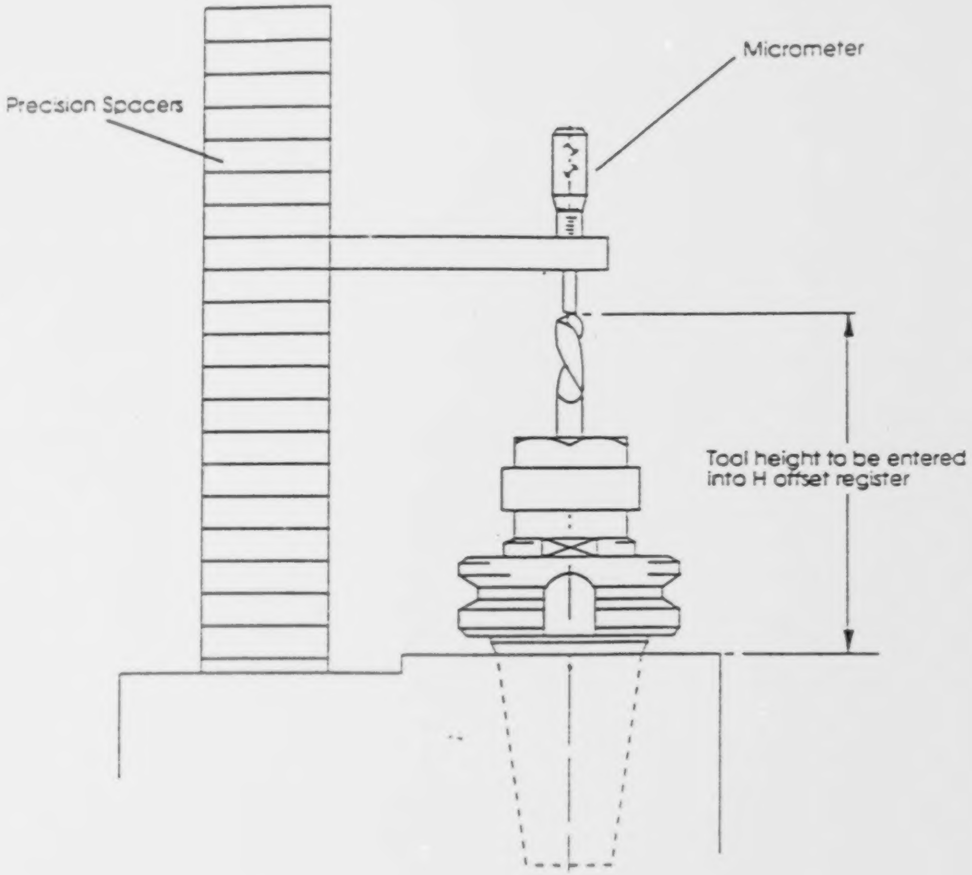
To determine the Z axis value, it is necessary to use a tool of known overall length (Gauge line - to tip). The distance travelled in the positional data display added onto the tool length, becomes the Z axis value.

This can also be entered manually into the work offset registers or within the program, e.g.

N10 G10 G90 L2 PO1 X-275.0 Y-300.0 Z-250.0

Once the datum positions have been recorded, it is necessary to measure the overall tool lengths which are to be used as offset values.

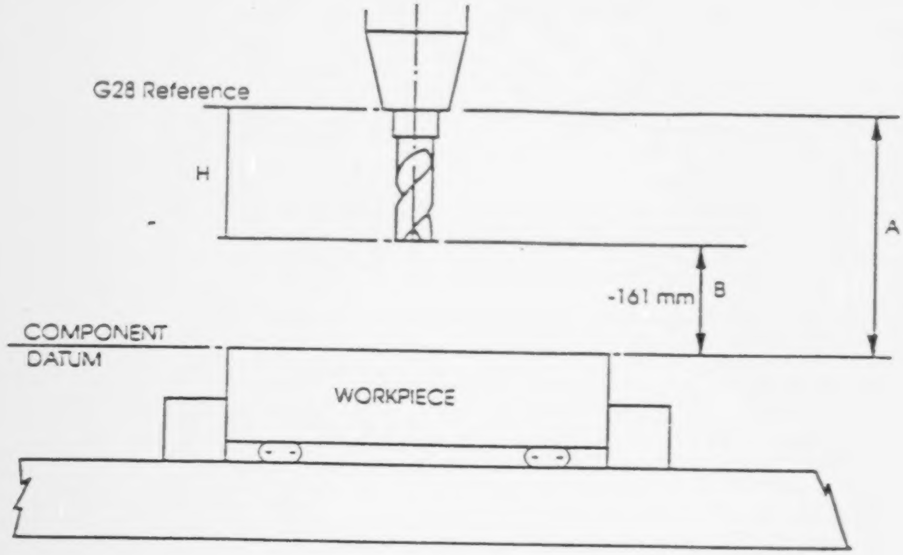
Methods of Measuring Tools


Precision Spacers
Micrometer
Tool height to be entered into H offset register

The tool to be measured is placed in the measuring system, and the tool height can be measured using the micrometer.

NOTE: This is a mechanical measuring system.

Methods of Measuring Tools


G28 Reference
H
A
-161 mm
B
COMPONENT
DATUM
WORKPIECE

With the Z axis positioned at the G28 Z axis reference position, we can determine the tool offset by using the formula:

H = A - B
Where H is the offset value
A is the distance from spindle gauge line to component datum
B is the distance between the tool tip and component datum

Example

H = A - B
H = 250 - 161.0
H = 89.0

NOTE: Each offset value found must be positive and entered into the appropriate offset number.

Your Notes:

Training Daily Guide

Session Two

During this two hour session, participants will:

1. Discuss toolholder manufacturing practices.
2. Examine NC toolholding systems.
3. Overview styles of toolholders.
4. Discuss styles of collets, tool adapters, and extensions.
5. Identify toolholders and various items.

Toolholder Manufacture

A. MATERIAL - 8620 ALLOY STEEL

B. HEAT TREAT - CARBURIZE AND HARDENED (Rc 58 EXTERIOR)
(Rc 35 INNER CORE)

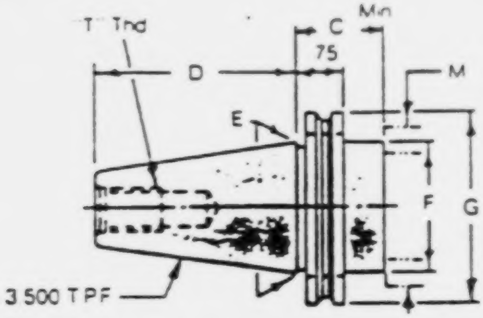
C. FINISH - BLACK OXIDE (NO BUILDUP ON OUTER SURFACE)

D. BLACK OXIDE - HELPS RELEASE TOOLHOLDER FROM THE SPINDLE

NC Toolholding Systems

1. TOOL HOLDER TYPES

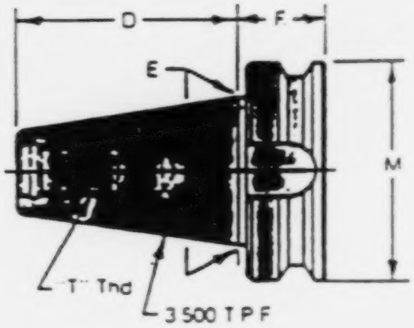
   A. V - FLANGE TOOLING (ALSO KNOWN AS CAT)
      1. V - 30
      2. V - 40 (POPULAR)
      3. V - 45
      4. V - 50 (POPULAR)



ENGINEERING DATA

taper	D	E	F	G	T
V-30	1.87	1.25	1.25	1.81	½-13 x 1.00
V-40	2.69	1.75	1.75	2.50	⅝-11 x 1.12
V-45	3.25	2.25	2.25	3.25	¾-10 x 1.50
V-50	4.00	2.75	2.75	3.88	1"-8 x 1.75

   B. BT TOOLING (JAPAN ORIGIN)
      1. BT - 35
      2. BT - 40 (POPULAR)
      3. BT - 45
      4. BT - 50 (POPULAR)

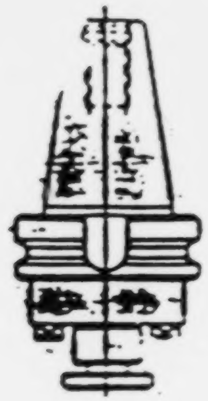


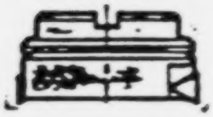
ENGINEERING DATA

taper	D	E	M	T	F
BT35	2.15	1.50	2.08	M12 x 1.75mm	.87
BT40	2.57	1.75	2.48	M16 x 2.00mm	1.06
BT45	3.25	2.25	3.34	M20 x 2.50mm	1.30
BT50	4.00	2.75	3.94	M24 x 3.00mm	1.50

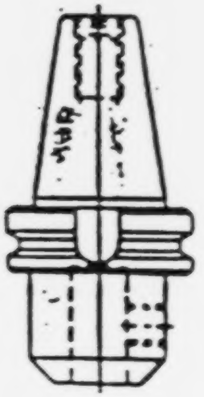
Styles of Toolholders

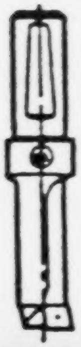
A. SHELL MILL ADAPTER - SHELL MILL

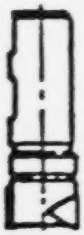


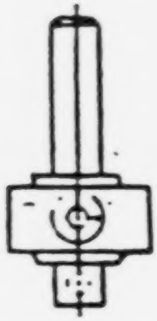


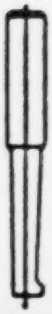
B. END MILL ADAPTER - END MILL, BORING CHUCK - BAR





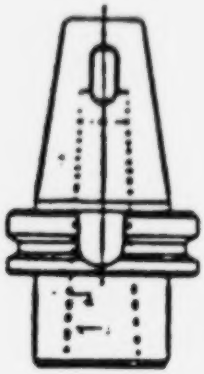




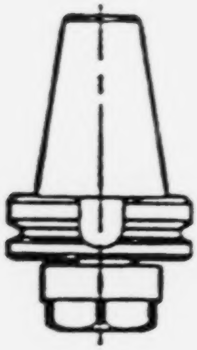


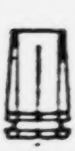
Styles of Toolholders

C. MORSE TAPER AND JACOBS TAPER



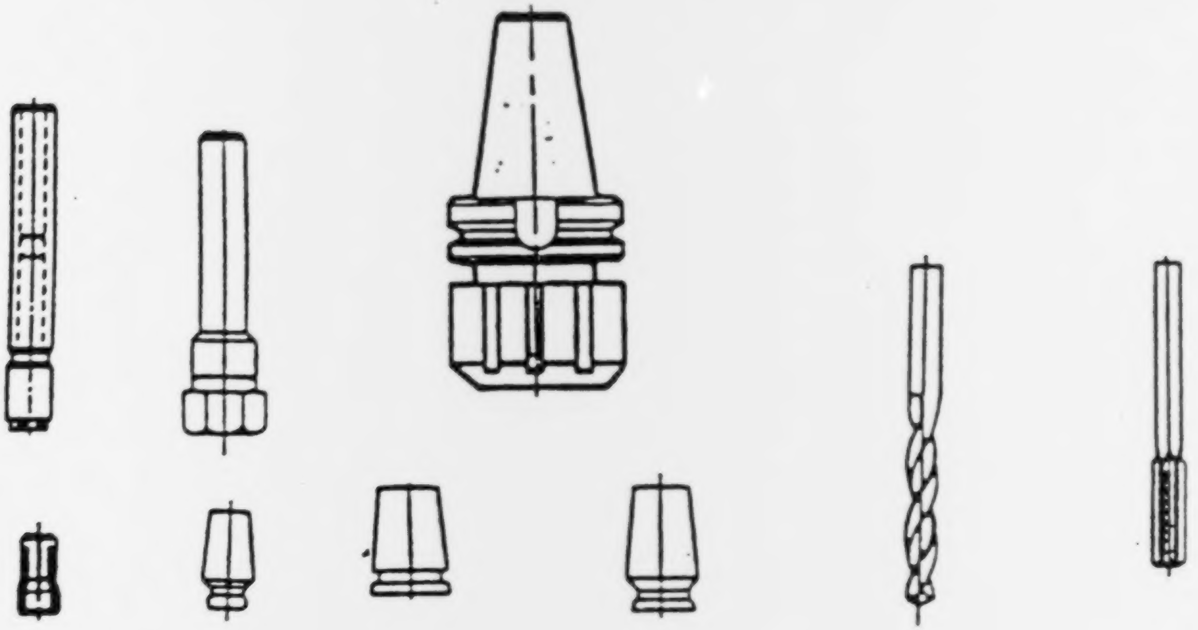
D. E - COLLET ADAPTER - E - COLLETS



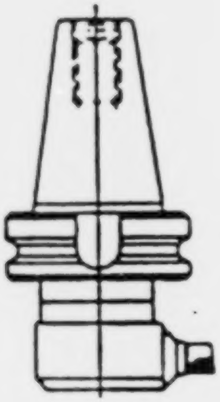


Styles of Toolholders

E. FLEX COLLET CHUCKS - GRIP & FLEX COLLETS



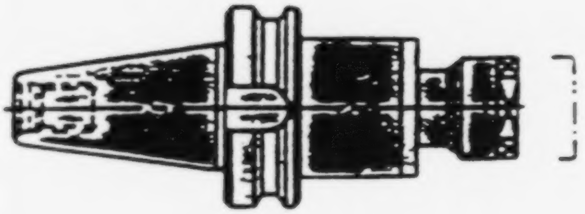
F. BORING RING ADAPTER



Styles of Toolholders

G. LENGTH COMPENSATING TAP CHUCK - TAP NUT / COLLET REQUIRED

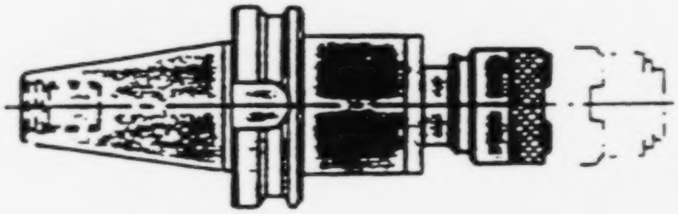
[.88 TENSION (EXPAND) & .38 COMPRESSION]

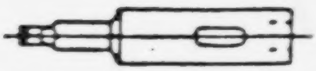


H. TAP COMPENSATING TAP ADAPTER - USE UNIVERSAL TAP ADAPTER

[.88 TENSION (EXPAND) & .38 COMPRESSION]

[SIZE 1 0 - 9/19 TAP & 1/8 PIPE]
[SIZE 2 5/16 - 7/8 TAP & 1/4 ,1/2 PIPE]
[SIZE 3 13/16 - 1-3/8 TAP & 3/4, 1" PIPE]





2.5 TOOLHOLDER IDENTIFICATION SYSTEM

C 4 E 4 --- 1 0 0 0

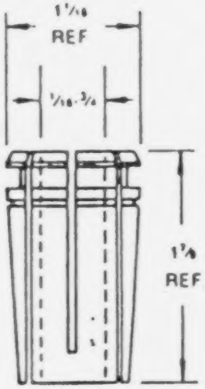
SHANK STYLE	TAPER	HOLDER	LENGTH	SIZE (DEC)
C = V-FLANGE	2 = 30 TAPER	A = AUTOMATIVE	2 = EXTRA SHORT	
B = BT FLANGE	3 = 35 TAPER	C = COLLET	3 = STUB	
M = MILACRON	4 = 40 TAPER	E = END MILL	4 = STANDARD	
	5 = 45 TAPER	F = STUB ARBOR	5 = EXTENDED	
	6 = 50 TAPER	H = BORING HEAD	6 = EXTRA EXTENDED	
		J = JACOBS TAPER		
		M = MORSE TAPER		
		P = RIGID TAP HOLDER		
		S = SHELL MILL		
		T = TAP		
		W = TEST BAR		
		X = BLANK BAR		

A. " E " COLLET (SINGLE ANGLE)

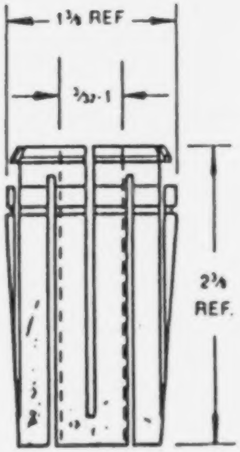
A. FITS ERICKSON STYLE COLLET CHUCKS

B. EACH SIZE HAS A GRIPPING RANGE OF 1/64 "

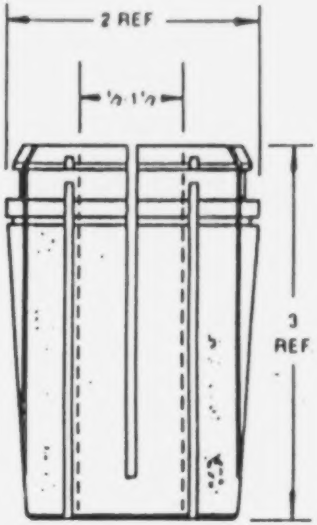
C. SERIES AVAILABLE 075, 100, AND 150


1 1/16
REF
1/16-3/4
1 1/8
REF

075 SERIES


1 3/8 REF
3/32-1
2 3/8
REF.

100 SERIES

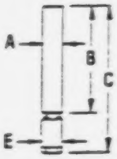

2 REF
1/2-1 1/2
3
REF.

150 SERIES

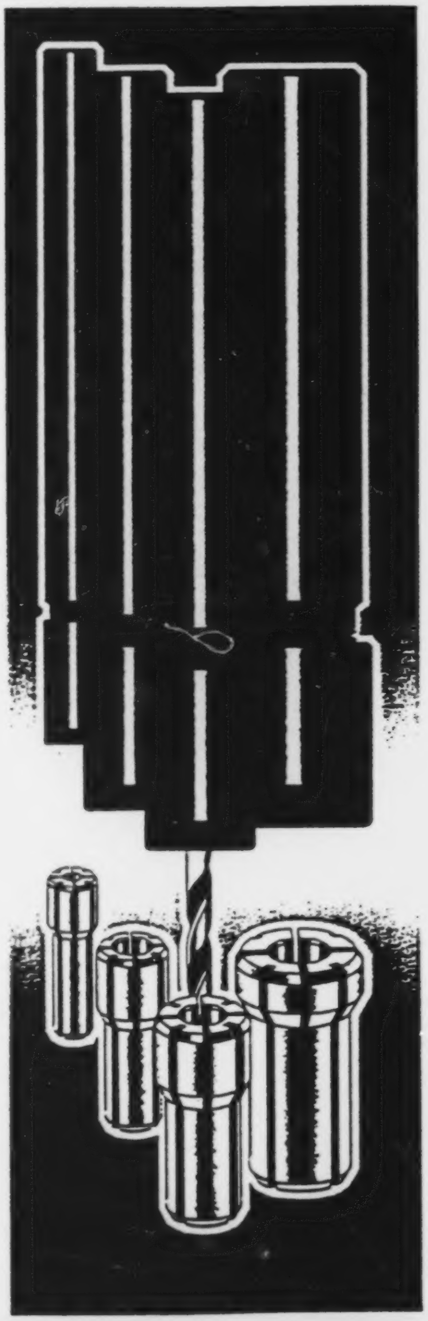
- Four different series to fit many needs
- Different nose diameters to solve clearance problems
- Collets collapse 1/64"
- Black oxide finish to protect precision ground surfaces
- The last four digits of our collet part numbers are the ID size

NOTE: For best grip and precision we recommend the Micro Precision™ Collet series as found on preceding pages

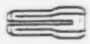
Command Double Angle Collet Extensions

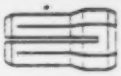


PART NO.	RANGE	COLLET	A	B	C	E
DACE-0030	.046-.250	DA30	.500	5.50	6.88	.56
DACE-0020	.062-.375	DA20	.750	5.50	7.23	.83
DACE-0010	.125-.562	DA10	1.000	5.50	7.50	1.06
DACE-0018	.250-.750	DA18	1.250	5.50	7.41	1.44

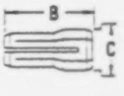


DOUBLE ANGLE COLLETS



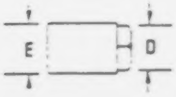


DA30	DA20	DA10		DA18	
PART NO.	PART NO.	PART NO.	PART NO.	PART NO.	PART NO.
DA30-0046	DA20-0062	DA10-0125	DA10-0375	DA18-0250	DA18-0531
DA30-0062	DA20-0078	DA10-0140	DA10-0390	DA18-0265	DA18-0546
DA30-0078	DA20-0093	DA10-0156	DA10-0406	DA18-0281	DA18-0562
DA30-0093	DA20-0109	DA10-0171	DA10-0421	DA18-0296	DA18-0578
DA30-0109	DA20-0125	DA10-0187	DA10-0437	DA18-0312	DA18-0593
DA30-0125	DA20-0140	DA10-0203	DA10-0453	DA18-0328	DA18-0609
DA30-0140	DA20-0156	DA10-0218	DA10-0468	DA18-0343	DA18-0625
DA30-0156	DA20-0171	DA10-0234	DA10-0484	DA18-0359	DA18-0640
DA30-0171	DA20-0187	DA10-0250	DA10-0500	DA18-0375	DA18-0656
DA30-0187	DA20-0203	DA10-0265	DA10-0515	DA18-0390	DA18-0671
DA30-0203	DA20-0218	DA10-0281	DA10-0531	DA18-0406	DA18-0687
DA30-0218	DA20-0234	DA10-0296	DA10-0546	DA18-0421	DA18-0703
DA30-0234	DA20-0250	DA10-0312	DA10-0562	DA18-0437	DA18-0718
DA30-0250	DA20-0265	DA10-0328	SET:	DA18-0453	DA18-0734
SET:	DA20-0281	DA10-0343	DA10-2900	DA18-0468	DA18-0750
DA30-1400	DA20-0296	DA10-0359		DA18-0484	SET:
	DA20-0312			DA18-0500	DA18-3300
	DA20-0328			DA18-0515	
	DA20-0343				
	DA20-0359				
	DA20-0375				
	SET:				
	DA20-2100				



COLLET	B	C
DA30	1.000	.375
DA20	1.188	.531
DA10	1.438	.769
DA18	1.625	1.035

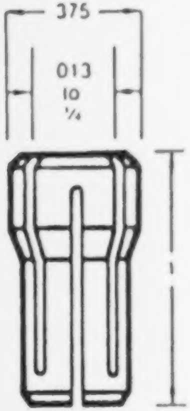
REPLACEMENT NUT



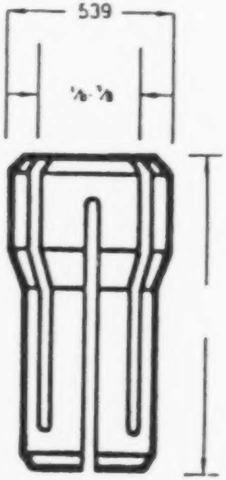
PART NO.	D	E
DACN-0030	1/2	.56
DACN-0020	3/4	.83
DACN-0010	31/32	1.06
DACN-0018	1 1/4	1.44

B. DOUBLE ANGLE " E " COLLET

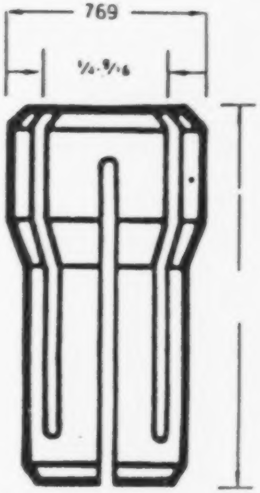
- A. FITS ERICKSON STYLE COLLET CHUCKS
- B. EACH SIZE HAS A GRIPPING RANGE OF 1/32 "
- C. SERIES AVAILABLE 300, 200, AND 100
- D. DESIGNED FOR HOLDING SMALL DIA. TOOLS
- E. NOT AS GOOD AS SINGLE ANGLE - SEATING PROBLEM
- F. CAPACITY OF 300 SERIES : .013 - .25
- G. CAPACITY OF 200 SERIES : .125 - .375
- H. CAPACITY OF 100 SERIES : .25 - .5625



300 SERIES

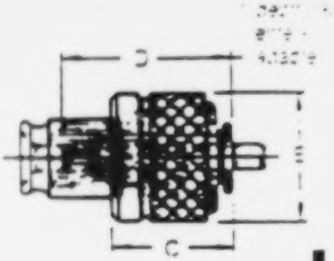


200 SERIES



100 SERIES

Tap Adapters & Extensions

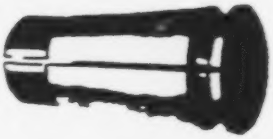


A. UNIVERSAL TAP ADAPTER

- A. POSITIVE DRIVE STYLE
- B. TORQUE CONTROL STYLE
    - 1. BEST USED FOR BLIND HOLES
    - 2. IF TORQUE SET TOO LIGHT , NOT ENOUGH THREADS
    - 3. TORQUE SETTINGS TRIAL AND ERROR

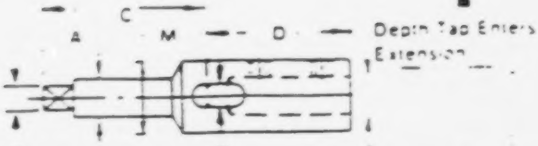
B. COLLET / NUT POSITIVE TAP DRIVE

- A. USE FOR RIGID TAPPING



B. TAP EXTENSION

- 1. POSITIVE TAP DRIVE
- 2. SEVERAL SIZES AVAILABLE

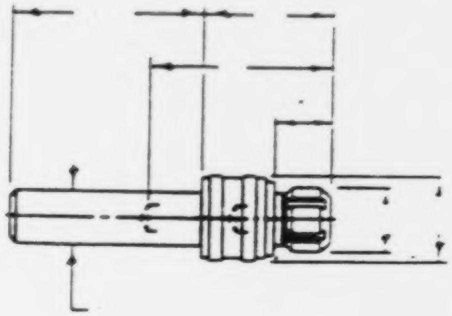

C
A
M
D
Depth Tap Enters Extension
B
B
A

Floating Reamer Holders

A. ALLOWS REAMER TO ALIGN TO PREVIOSLY DRILLED HOLE

B. MOST ALLOW .06 TOTAL RADIAL FLOAT

C. TOOLHOLDER AND COLLET TO MATCH IN SIZE CAPACITY

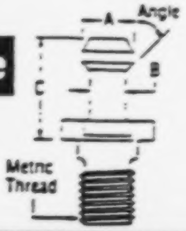


Toolholder Retention Stud

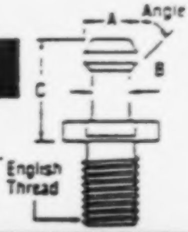
6. TOOLHOLDER RETENTION STUD (RETENTION KNOB)

A. TOOLHOLDER RETENTION STUDS ARE NOT NECESSARILY INTERCHANGEABLE ACROSS VARIOUS TYPES OF TOOLHOLDERS OR MACHINE TOOL MODELS.

BT-Flange



V-Flange



JAPANESE STYLE RETENTION KNOBS—

TAPER	PART NO.	A	B	C	ANGLE
30	RB2M-0001	.43	.27	.90	45°
	RB2M-0002				60°
35	RB3M-0001	.51	.33	1.10	45°
	RB3M-0100	.54	.35	1.10	60°
	RB3M-0101	.55	.31	.91	90°
40	RB4M-0001	.59	.39	1.38	45°
	RB4M-0002				60°
	RB4M-0003				90°
45	RB5M-0001	.75	.55	1.57	45°
	RB5M-0002				60°
	RB5M-0003				90°
50	RB6M-0001	.91	.67	1.78	45°
	RB6M-0002				60°
	RB6M-0003				90°

TAPER	PART NO.	A	B	C	ANGLE
40	RB4E-0001	.59	.39	1.27	45°
	RB4E-0002				60°
	RB4E-0003				90°
45	RB5E-0001	.75	.55	1.58	45°
	RB5E-0003				90°
50	RB6E-0001	.91	.67	1.78	45°
	RB6E-0002				60°
	RB6E-0003				90°
	RB6E-0001-C				45°
	RB6E-0002-C				60°

OTHER RETENTION KNOBS FOR DOMESTIC AND IMPORTED MACHINES

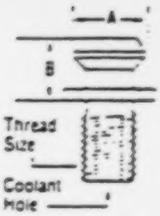
30	RB2M-0100	.47	.31	.94	R*
50	RS6M-0114-C	1.10	.83	1.34	75°

40	RB4E-0100	.59	.39	1.05	90°
	RB4E-0101	.59	.39	.98	90°
	RS4E-0104	.59	41	1.44	45°
	RS4E-0106	.59	.39	1.38	45°
	RS4E-0109	.52	.39	.47	50°
	RS4E-0110	.93	.83	.39	75°
	RS4E-0111	.50	.31	.75	75°
	RS4E-0114-C	.75	.55	1.02	75°
50	RB6E-0100**	.91	.67	1.78	90°
	RS6E-0101	.94	.71	1.23	90°
	RS6E-0101-C	.94	.71	1.23	90°
	RS6E-0114-C	1.10	.83	1.34	75°

* R designates radius. ** Wrench flats are on knob end.
C at end of Part No designates coolant hole.

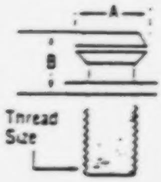
Flanges

V-Flange


A
B
Thread Size
Coolant Hole

U.S. ANSI — for machines *WITH* Thru-The-Spindle Coolant

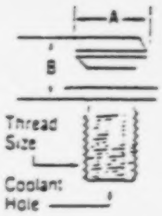
TAPER	PART NO.	A	B	Thread Size
30	RC2E-0001	.52	.76	1/2-13
40	RC4E-0001	.74	.64	5/8-11
45	RC5E-0001	.94	.82	3/4-10
50	RC6E-0001	1.14	1.00	1-8


A
B
Thread Size

U.S. ANSI — for machines *WITHOUT* Thru-The-Spindle Coolant

TAPER	PART NO.	A	B	Thread Size
40	RC4E-0002	.74	.64	5/8-11
45	RC5E-0002	.94	.82	3/4-10
50	RC6E-0002	1.14	1.00	1-8

BT-Flange


A
B
Thread Size
Coolant Hole

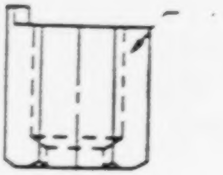
MODIFIED U.S. ANSI — with metric thread and coolant hole

TAPER	PART NO.	A	B	Thread Size
40	RS4M-0105	.74	.64	M16-2
	RS4M-0115	.74	.75	M16-2
45	RS5M-0106	.94	.82	M20-2.5
50	RS6M-0105	1.14	1.00	M24-3.0

Adapter Locking Fixture

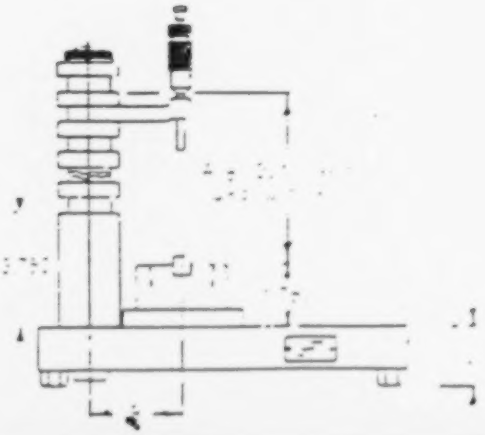
A. HOLDS AND PROTECTS THE ADAPTER SHANK WHEN CHANGING TOOLS IN COLLET OR SHELL MILL ADAPTERS.

B. V - 30 AND BT - 35	2.5 INCH HEX SHANK ON FIXTURE
C. V - 40 AND BT - 40	3.0 INCH HEX SHANK ON FIXTURE
D. V - 45 AND BT - 45	3.5 INCH HEX SHANK ON FIXTURE
E. V - 50 AND BT - 50	4.0 INCH HEX SHANK ON FIXTURE



8. PRE-SETTING HEIGHT GAGE

A. ALLOWS FOR SETTING OF TOOL LENGTHS OFF OF MACHINE.



Identify the Toolholders & Items

A ____

B ____

C ____

D ____

E ____

F ____

G ____

H ____

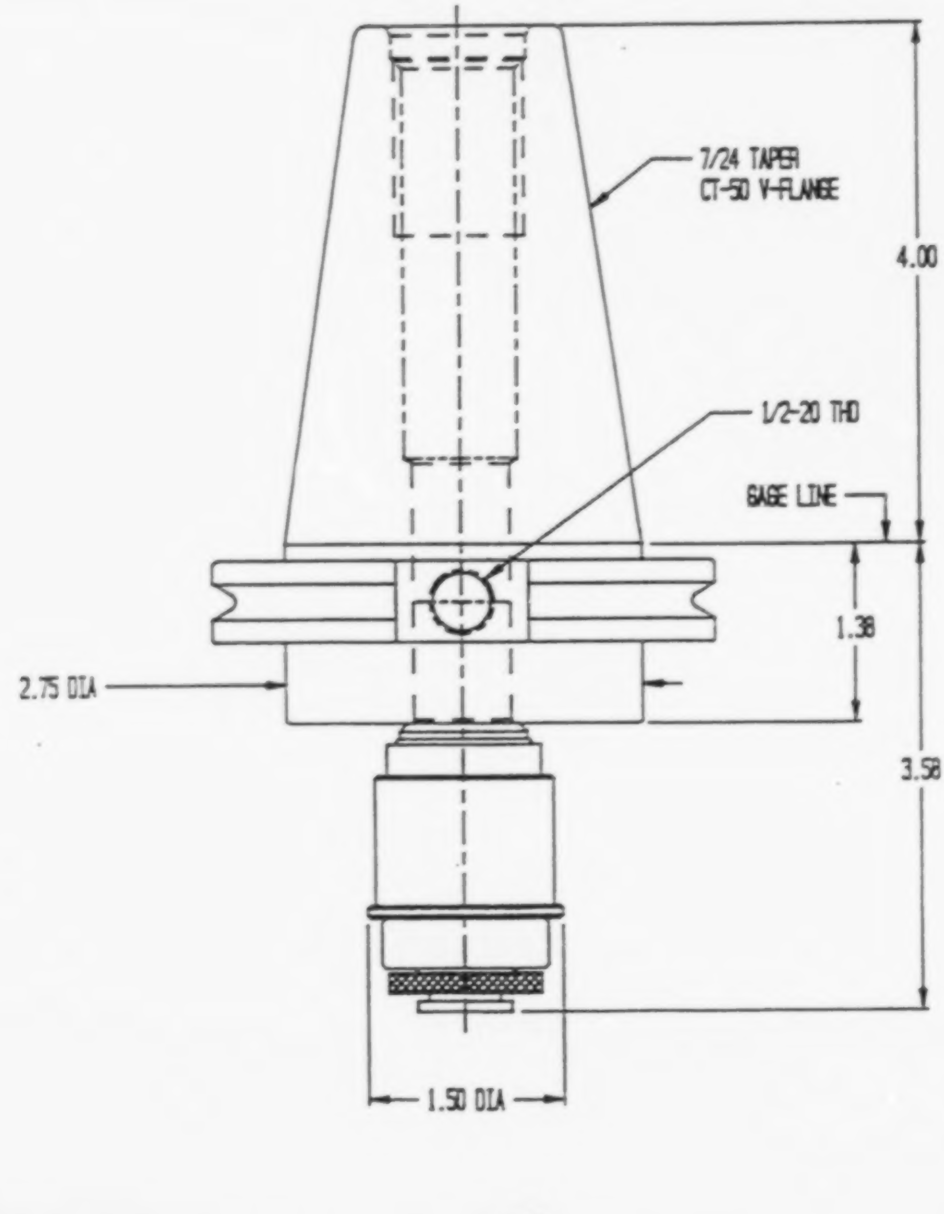
I ____

Command Toolholder Blueprints

COMMAND

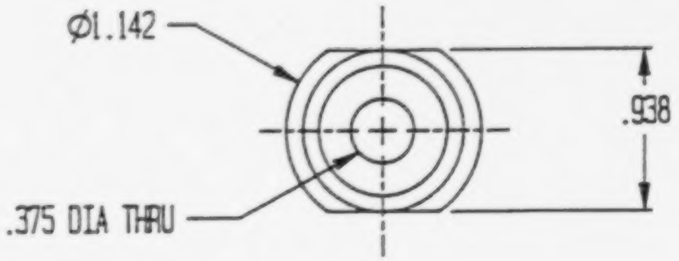
DESCRIPTION: CT-50, .750 I.D. ADAPTOR SHANK
ASSEMBLED WITH XRTH-0001

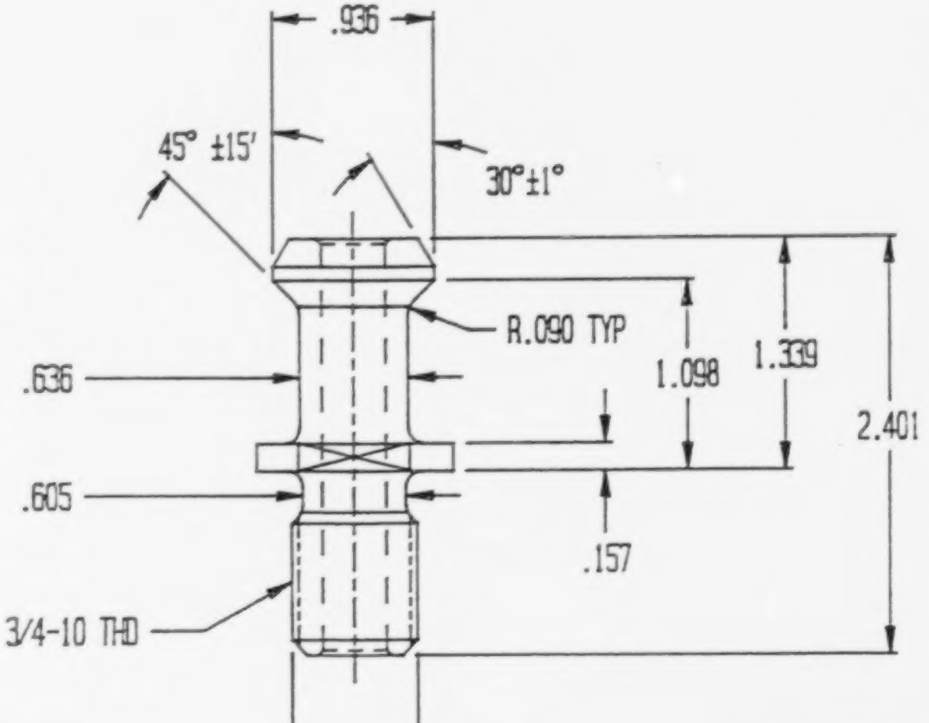
DATE	SCALE	DRAWN	SHEET	PRT NO: C6P2-0001
5-27-93	.75	B.H.	1 OF 1	DWG NO: 122700-1


7/24 TAPER
CT-50 V-FLANGE
4.00
1/2-20 THD
GAGE LINE
1.38
2.75 DIA
3.58
1.50 DIA

COMMAND

DESCRIPTION: CT-45 RETENTION KNOB				MAT'L: 8620 CARB. .02-.03 DP DRAW 58-60 RC
DATE	SCALE	DRAWN	SHEET	PRT NO: RSSE-0205-C
3-9-94	1-1	WRK	1 OF 1	DWG NO: 6183-A


Ø1.142
.938
.375 DIA THRU

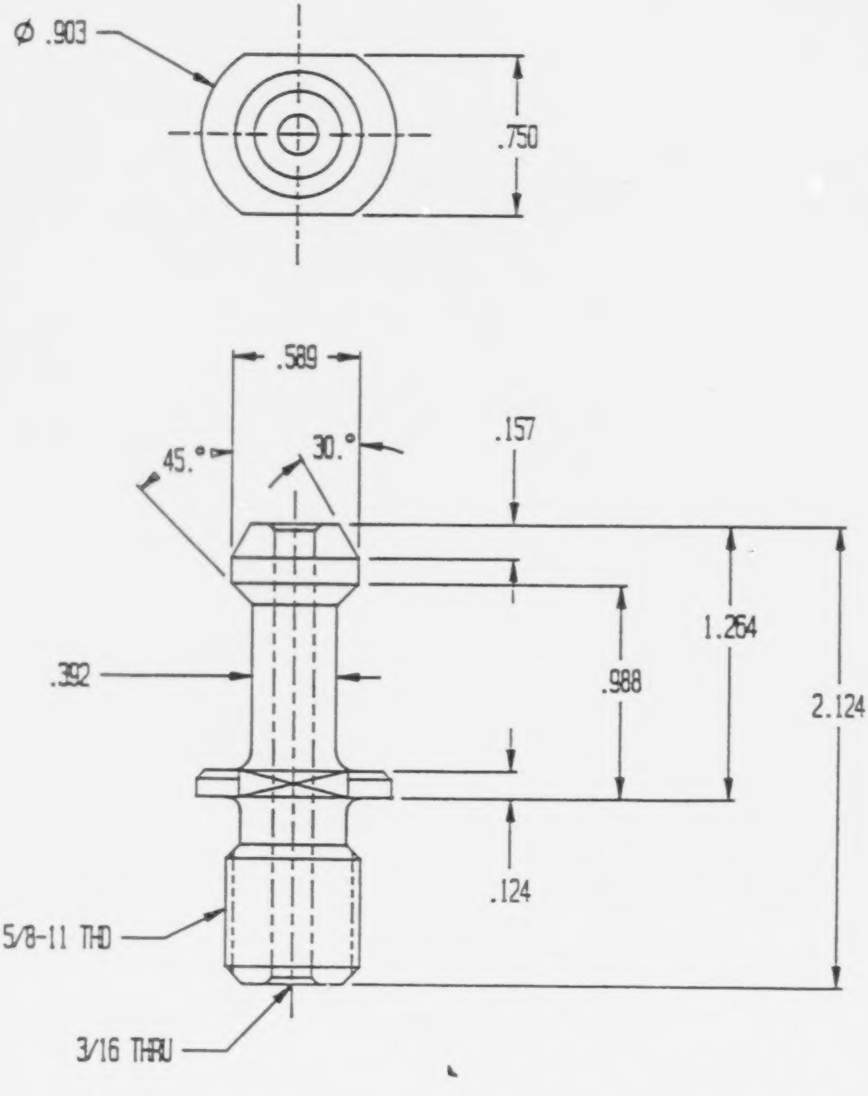

.936
45° ±15'
30°±1°
R.090 TYP
.636
1.098
1.339
2.401
.605
.157
3/4-10 THD
.735

TOLERANCES U.O.S.	
.X	±.030
.XX	±.020
.XXX	±.010
.XXXX	±.005
ANGLES	±1 DEG

COMMAND

DESCRIPTION: BT-40 RETENTION KNOB
WITH 3/16 COOLANT HOLE

DATE	SCALE	DRAWN	SHEET	PRT NO: RB4E-0001-C
7-1-93	1.25	B.H.	1 OF 1	DWG NO: 6152-A

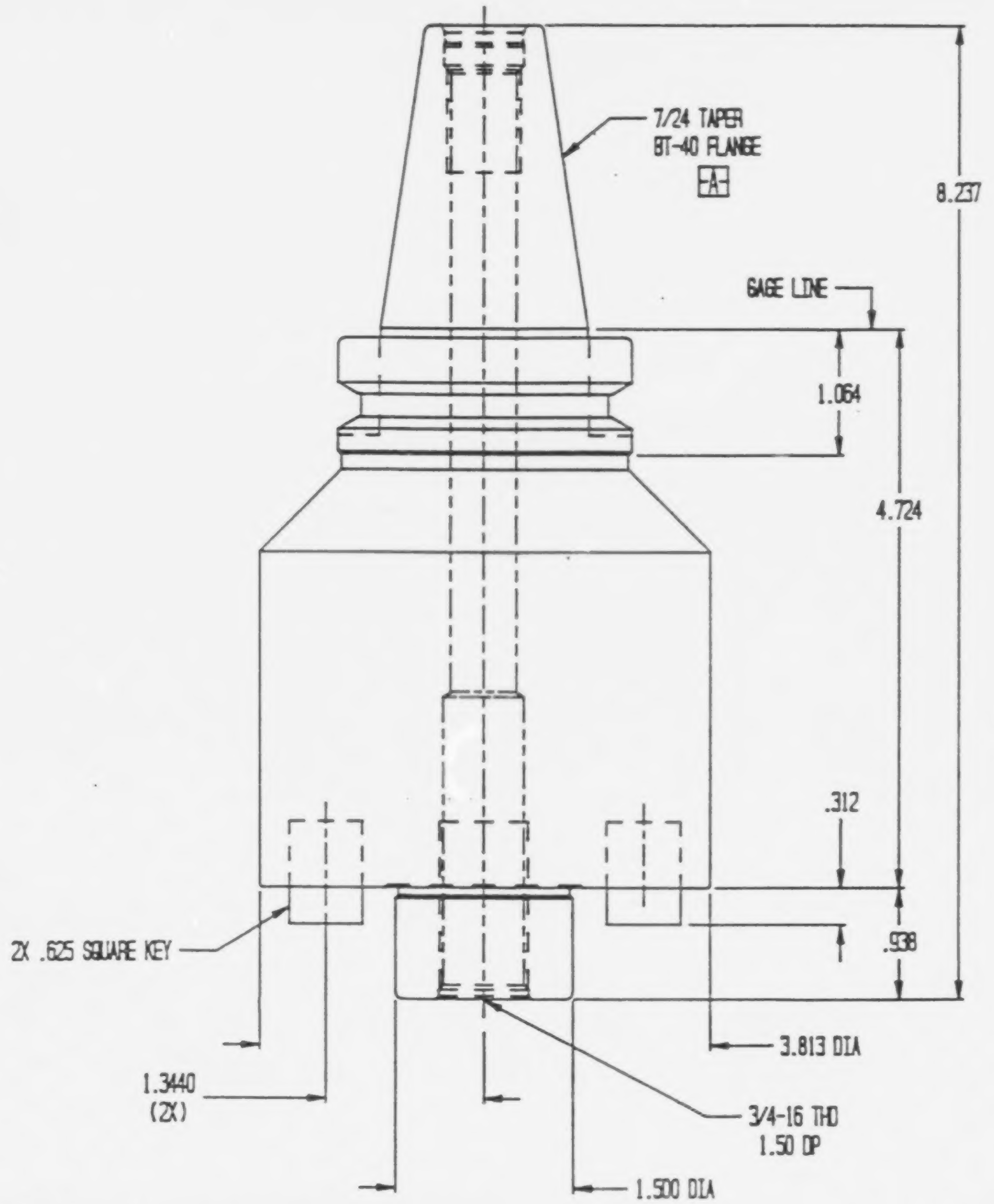


Ø .903
.750
.589
45.°
30.°
.157
1.264
.392
.988
2.124
.124
5/8-11 THD
3/16 THRU

COMMAND

DESCRIPTION: BT-40, EXTENDED LENGTH
1.500 SHELL MILL HOLDER

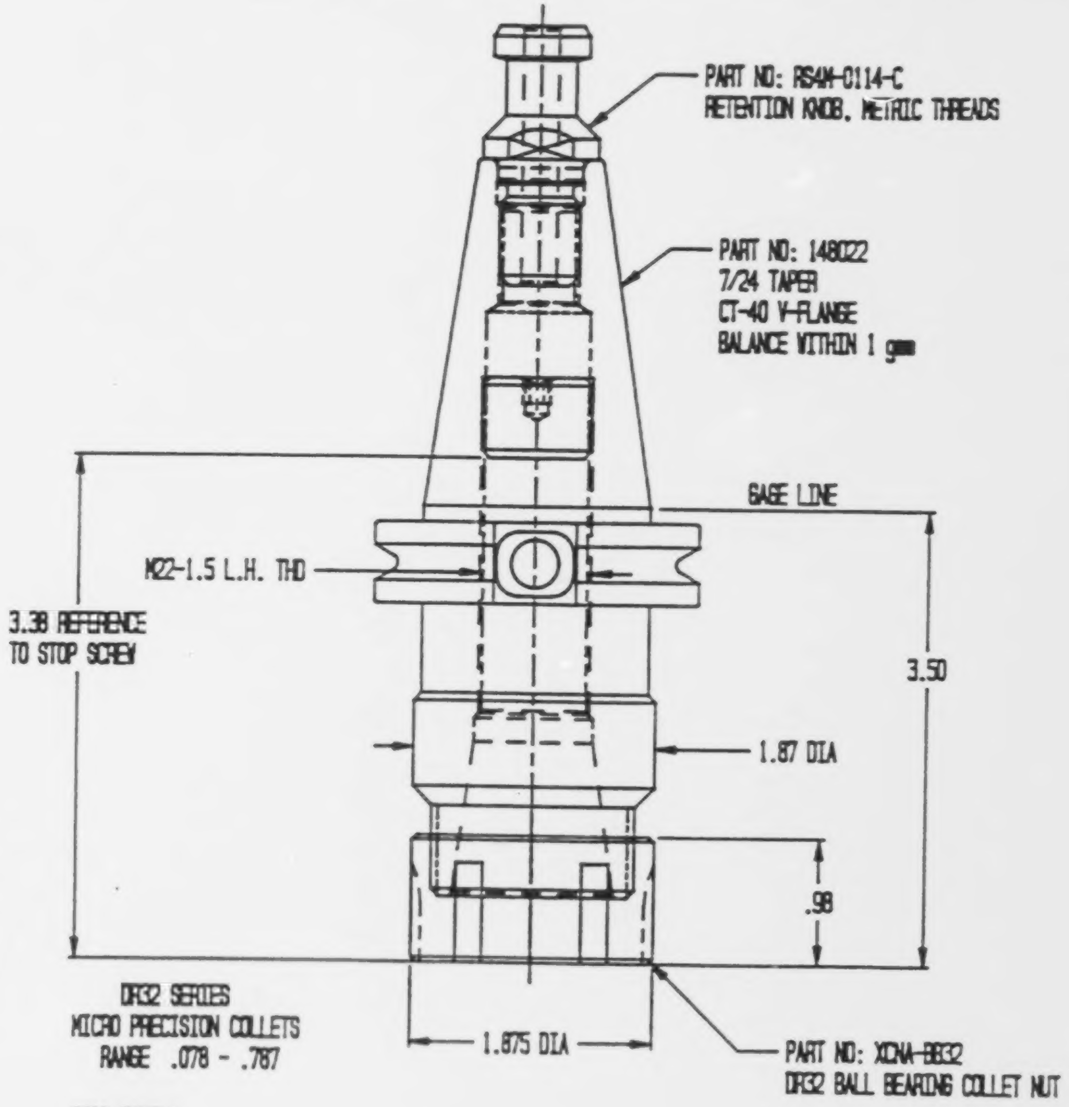
DATE	SCALE	DRAWN	SHEET	PRT NO: B4S5-1500
8-29-92	.75	B.H.	1 OF 1	DWG NO: 242140-A



7/24 TAPER
BT-40 FLANGE
-A-
8.237
GAGE LINE
1.064
4.724
.312
.938
2X .625 SQUARE KEY
3.813 DIA
1.3440
(2X)
3/4-16 THD
1.50 DP
1.500 DIA

COMPONENT PARTS: XSDK-1500 - DRIVE KEY, 2 REQUIRED
XSAS-1500 - ARBOR SCREW, 3/4-16 THD

COMMAND	DESCRIPTION: CT-40 WITH BT-40 BACK HOLE & THD DR32 COLLET CHUCK NOSE LENGTH = 3.50 ASSEMBLED			ASSEMBLY DRAWING REFERENCE ONLY
	DATE	SCALE	DRAWN	SHEET	PRT NO: SPECIAL
	12-15-93	.75	B.H.	1 OF 1	DWG NO: 148022-A

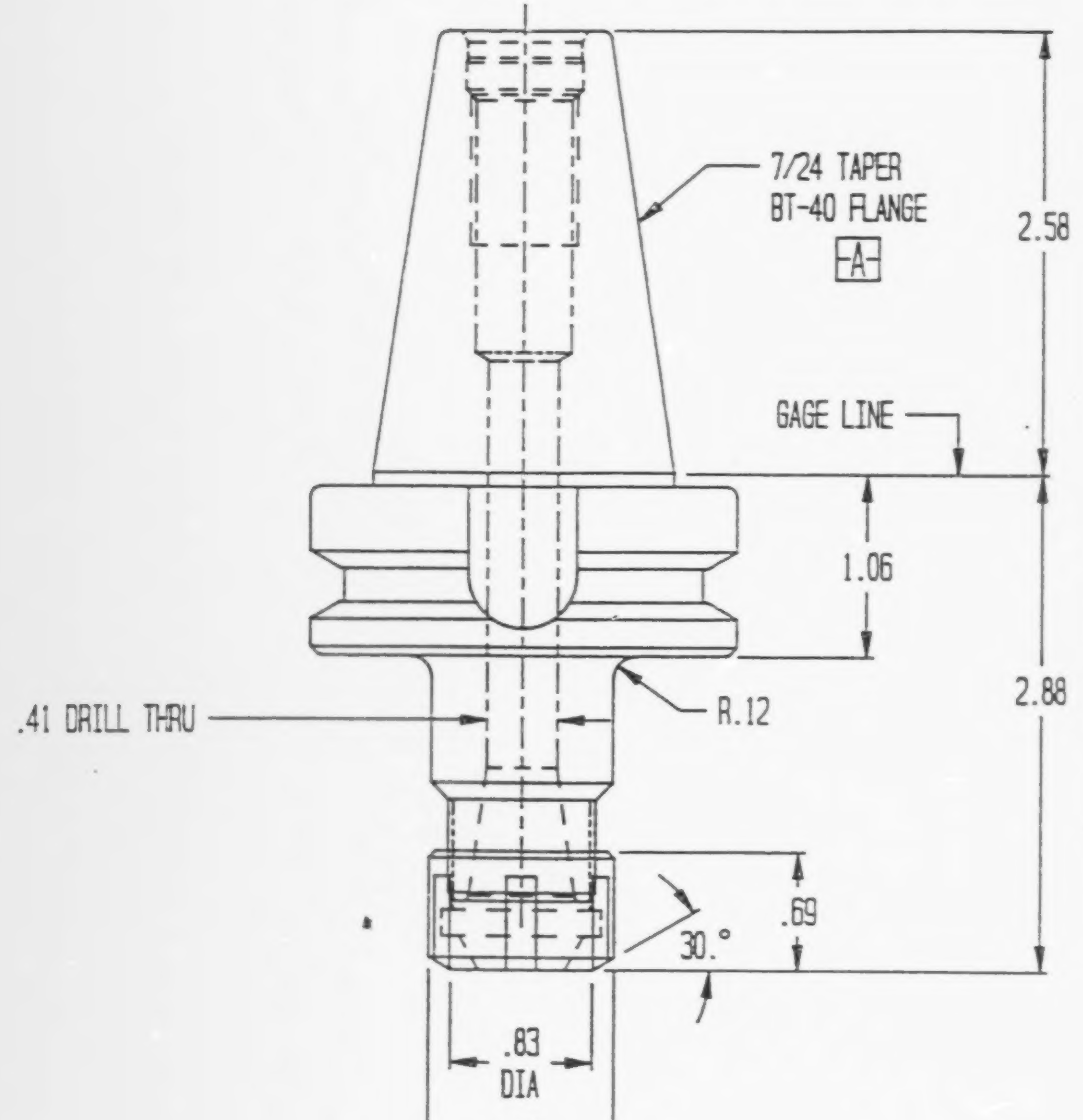

PART NO: RS4M-0114-C
RETENTION KNOB, METRIC THREADS
PART NO: 148022
7/24 TAPER
CT-40 V-FLANGE
BALANCE WITHIN 1 gmm
GAGE LINE
M22-1.5 L.H. THD
3.38 REFERENCE
TO STOP SCREW
3.50
1.87 DIA
.98
1.875 DIA
PART NO: XCNA-BB32
DR32 BALL BEARING COLLET NUT

DR32 SERIES
MICRO PRECISION COLLETS
RANGE .078 - .787

STOP SCREWS
PART NO: XCCS-2215
XCCS-2215-C (COOLANT,.12 THRU HOLE AND URETHANE WASHER INCLUDED)

TOLERANCES U.O.S.	
.X	±.030
.XX	±.020
.XXX	±.010
.XXXX	±.005
ANGLES	±1 DEG

COMMAND	DESCRIPTION: BT-40,STANDARD LENGTH DR16 COLLET CHUCK NOSE LENGTH=2.88" ASSEMBLED				
	DATE	SCALE	DRAWN	SHEET	PRT NO: B4C4-0016
	6-29-92	1-1	B.H.	1 OF 1	DWG NO: 241110-A

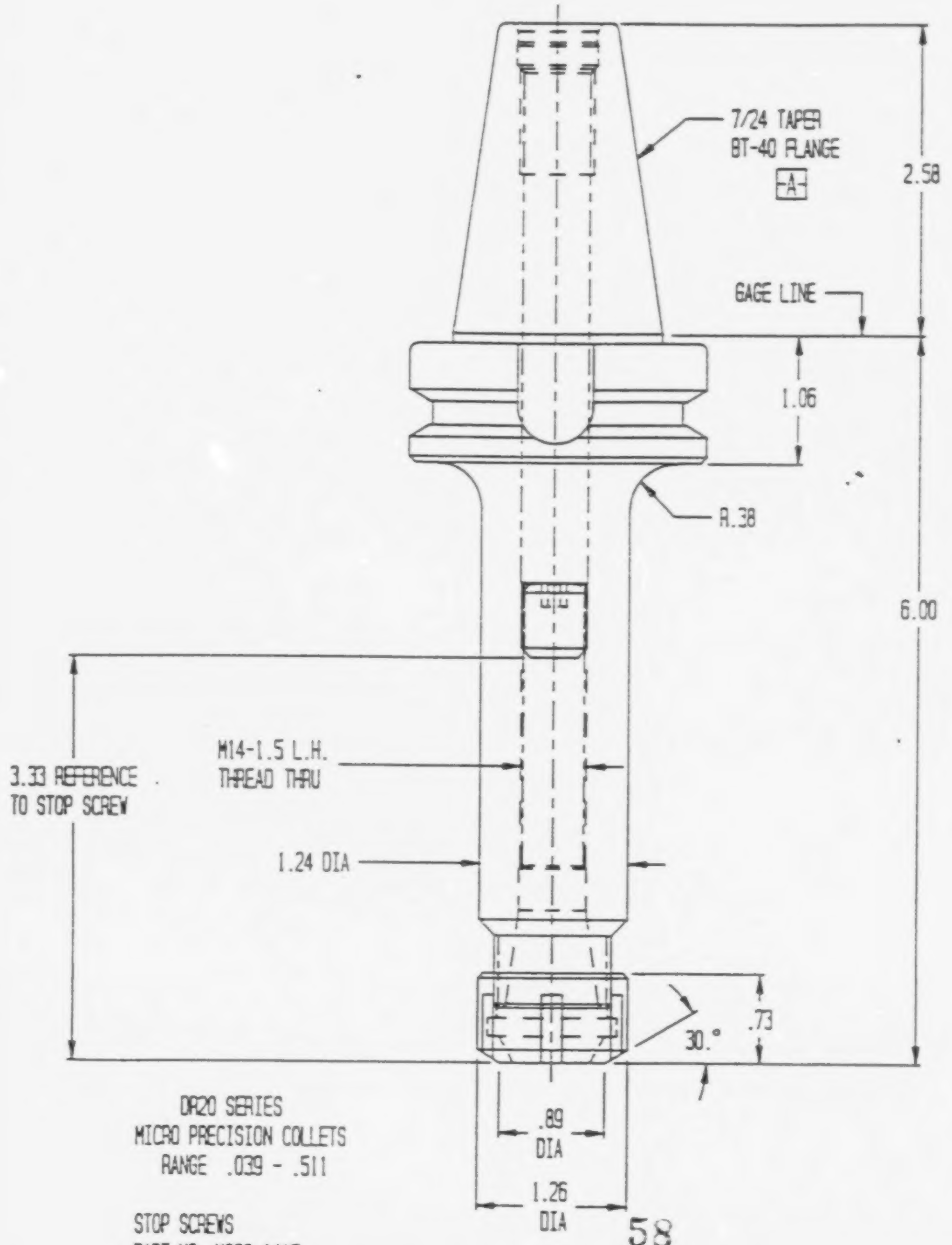

7/24 TAPER
BT-40 FLANGE
-A-
2.58
GAGE LINE
1.06
2.88
.41 DRILL THRU
R.12
.69
30.°
.83
DIA
1.08
DIA

DR16 SERIES
MICRO PRECISION COLLETS
RANGE .019 - .393

COMMAND

DESCRIPTION: BT-40, EXTENDED LENGTH
DR20 COLLET CHUCK
NOSE LENGTH=6.0" ASSEMBLED

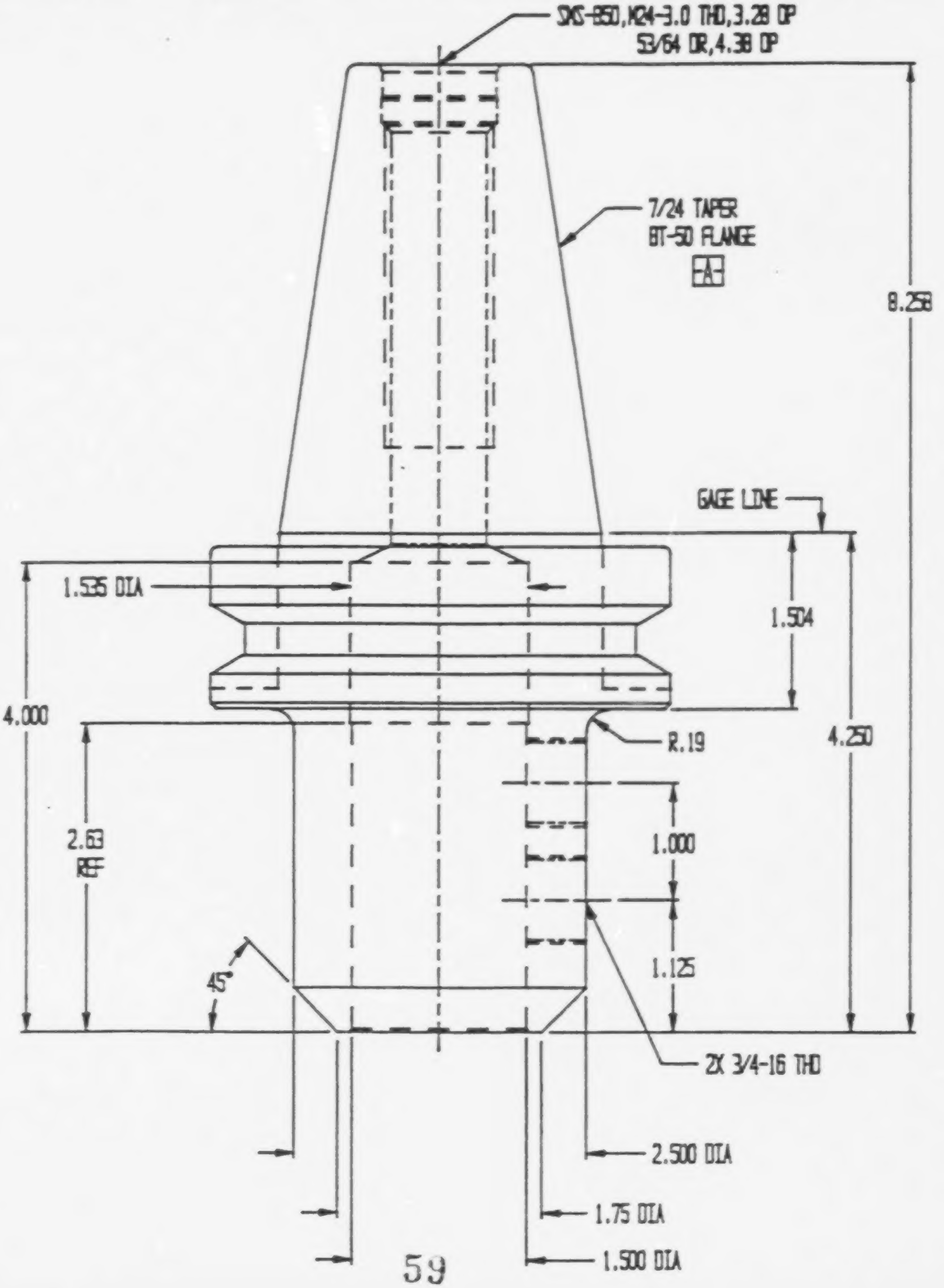
DATE	SCALE	DRAWN	SHEET	PRT NO: 84C5-0020
6-29-92	.75	B.H.	1 OF 1	DWG NO: 241150-A


7/24 TAPER
BT-40 FLANGE
-A-
2.58
GAGE LINE
1.06
R.38
6.00
3.33 REFERENCE
TO STOP SCREW
M14-1.5 L.H.
THREAD THRU
1.24 DIA
30.°
.73
.89
DIA
1.26
DIA

DR20 SERIES
MICRO PRECISION COLLETS
RANGE .039 - .511

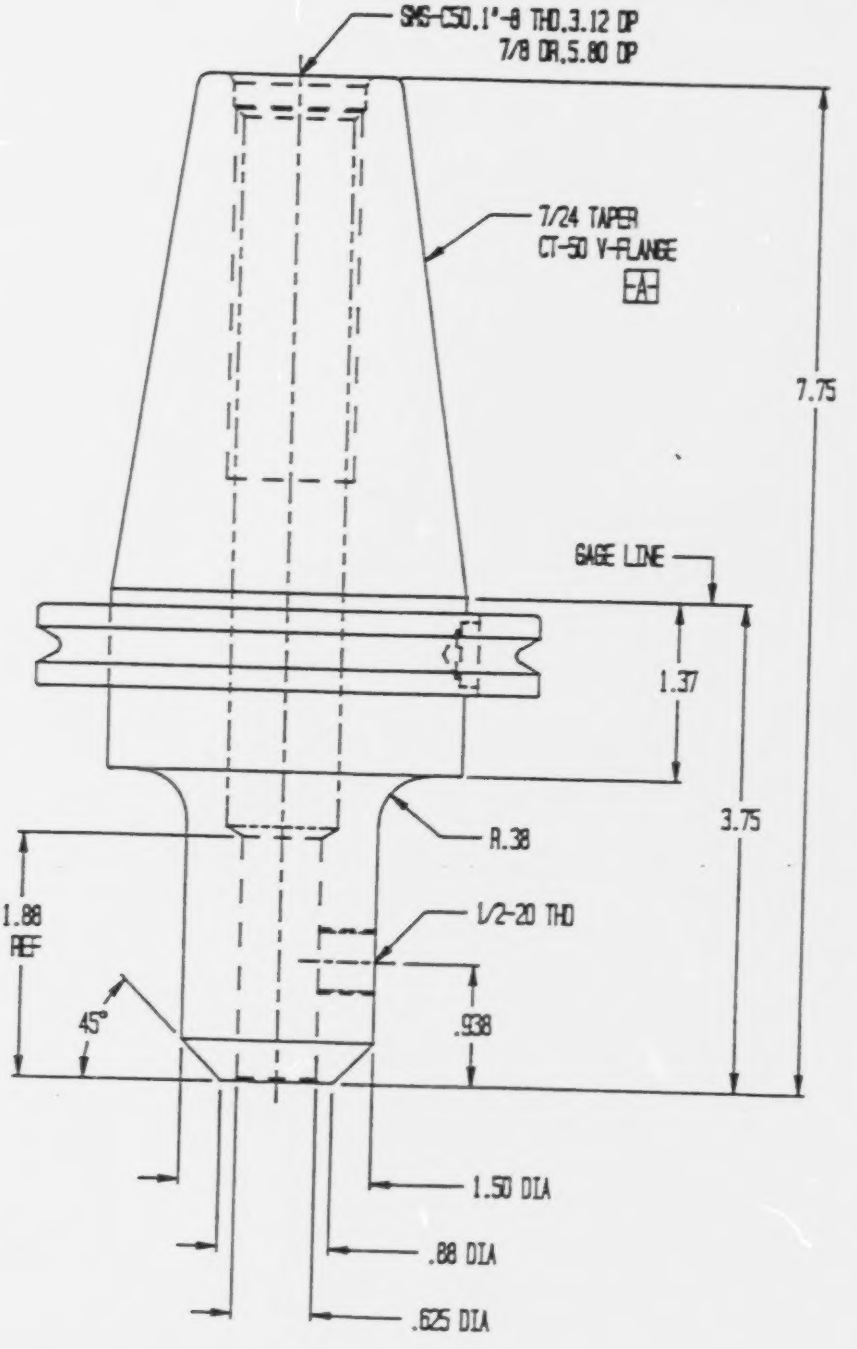
STOP SCREWS
PART NO: XCCS-1415
XCCS-1415-C (COOLANT, .12 THRU HOLE AND URETHANE WASHER INCLUDED)

COMMAND	DESCRIPTION: BT-50, STANDARD LENGTH 1.500 END MILL HOLDER			MAT'L:
DATE	SCALE	DRAWN	SHEET	PRT NO: B6E4-1500
3/7/95	.75	R.J.	1 OF 1	DWG NO: 221760-A


SKS-B50,M24-3.0 THD,3.28 DP
53/64 DR,4.38 DP
7/24 TAPER
BT-50 FLANGE
-A-
8.258
GAGE LINE
1.535 DIA
1.504
4.000
R.19
4.250
2.63
REF
1.000
1.125
45°
2X 3/4-16 THD
2.500 DIA
1.75 DIA
1.500 DIA

COMMAND

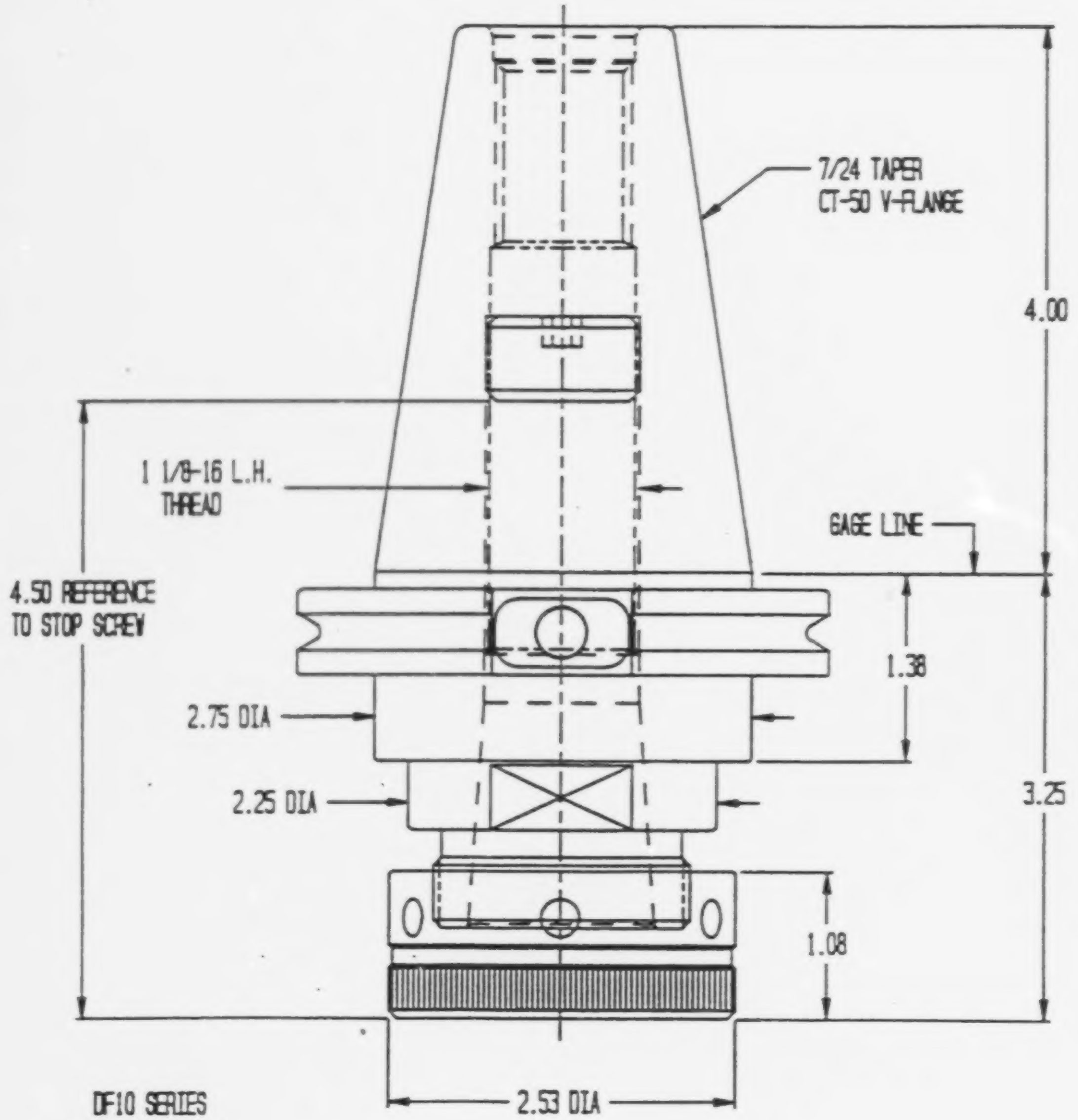
DESCRIPTION: CT-50, STANDARD LENGTH .625 END MILL HOLDER				
DATE	SCALE	DRAWN	SHEET	PRT NO: C6E4-0625
6-4-93	.75	B.H.	1 OF 1	DWG NO: 121610-A


SMS-C50.1"-8 THD.3.12 DP
7/8 DR.5.80 DP
7/24 TAPER
CT-50 V-FLANGE
EA
7.75
GAGE LINE
1.37
3.75
R.38
1.88
REF
1/2-20 THD
45°
.938
1.50 DIA
.88 DIA
.625 DIA

COMMAND

DESCRIPTION: CT-50,STANDARD LENGTH
DF10 COLLET CHUCK
NOSE LENGTH=3.25" ASSEMBLED

DATE	SCALE	DRAWN	SHEET	PRT NO: C6C4-1000
9-18-91	.80	B.H.	1 OF 1	DWG NO: 121320-A


7/24 TAPER
CT-50 V-FLANGE
4.00
1 1/8-16 L.H.
THREAD
GAGE LINE
4.50 REFERENCE
TO STOP SCREW
1.38
2.75 DIA
2.25 DIA
3.25
1.08
2.53 DIA

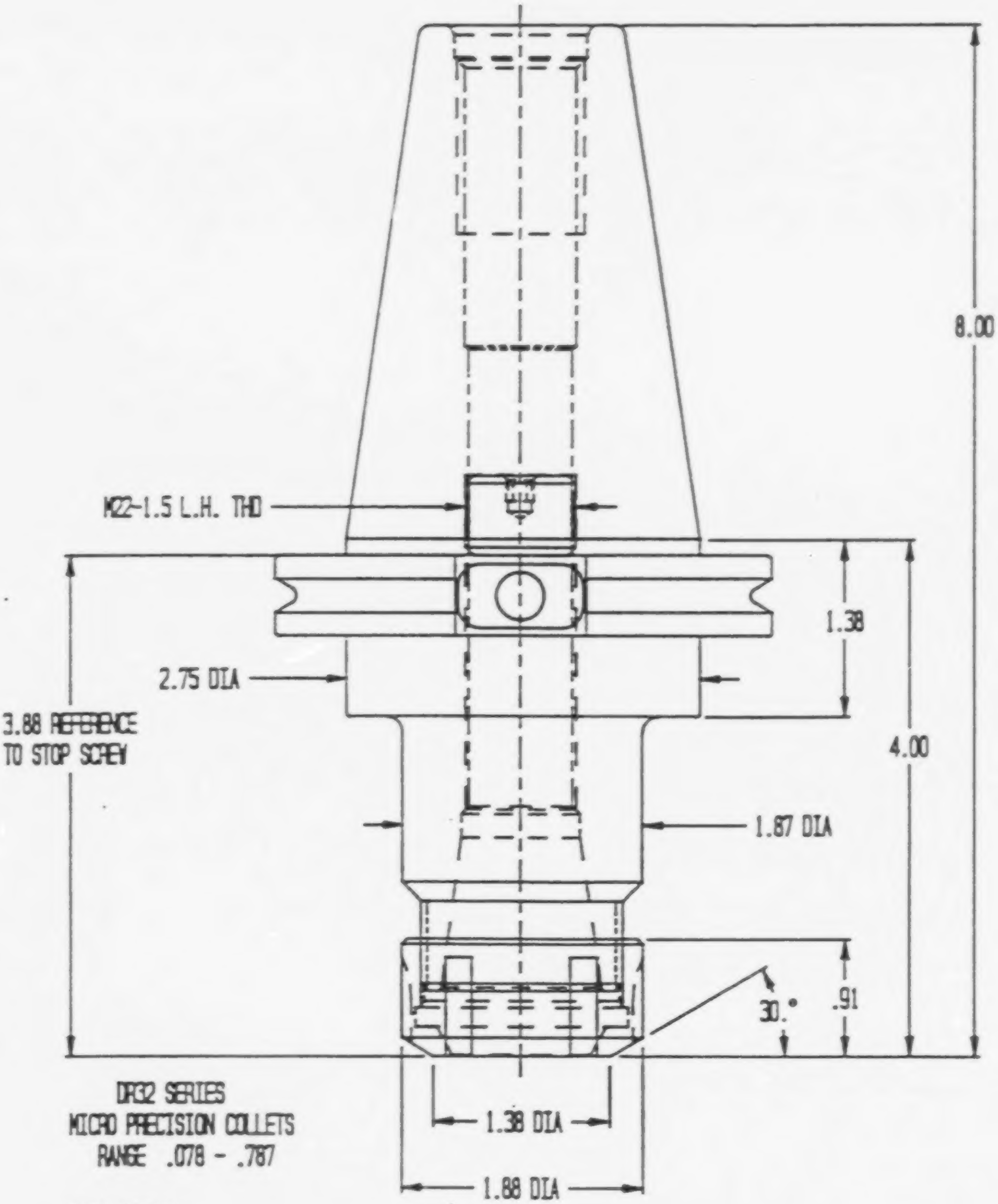
DF10 SERIES
COMFLEX COLLETS
RANGE .094 - 1.000

STOP SCREWS
PART NO: XCCS-1816
XCCS-1816-C (COOLANT,.25 THRU HOLE AND URETHANE WASHER INCLUDED)

COMMAND

DESCRIPTION: CT-50, STANDARD LENGTH
DR32 COLLET CHUCK
NOSE LENGTH=4.0" ASSEMBLED

DATE	SCALE	DRAWN	SHEET	PRT NO: C6C4-0032
6-4-93	.75	B.H.	1 OF 1	DWG NO: 121210-A


M22-1.5 L.H. THD
8.00
1.38
2.75 DIA
3.88 REFERENCE
TO STOP SCREW
4.00
1.87 DIA
30.°
.91
1.38 DIA
1.88 DIA

DR32 SERIES
MICRO PRECISION COLLETS
RANGE .078 - .787

STOP SCREWS
PART NO: XCCS-2215
XCCS-2215-C (COOLANT, .12 THRU HOLE AND URETHANE WASHER INCLUDED)

Your Notes:

Your Notes:

Sections Three & Four

Your Notes:

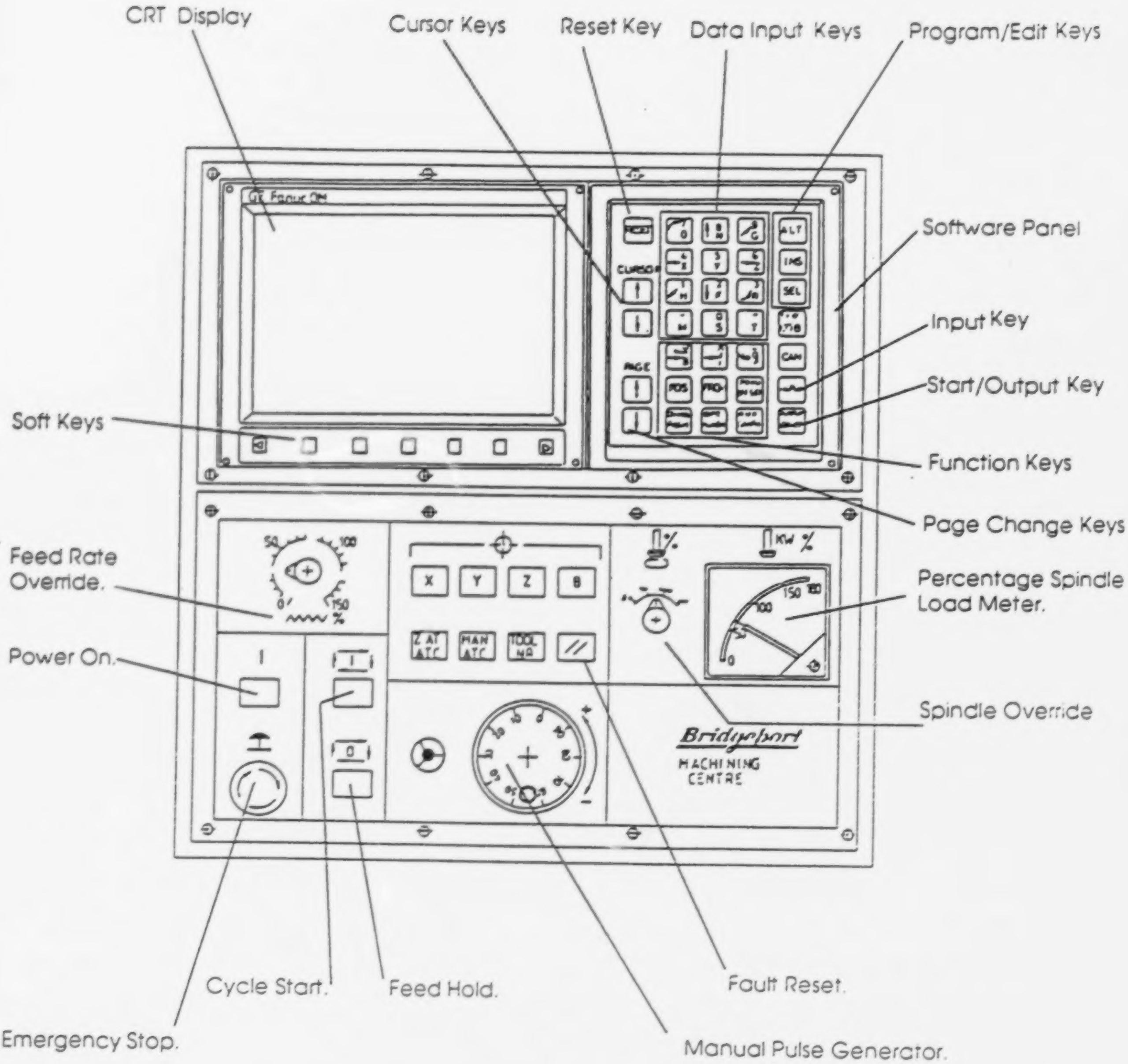
Training Daily Guide

Sessions Three & Four

During this four hour segment, participants will:

1. **Discuss functions and keys as found on a CNC Operator Panel.**
2. **Execute a short MDi command using appropriate keys.**
3. **"Write" a short CNC drill program using data input keys.**
4. **"Write" a short CNC milling program using data input keys.**
5. **"Edit" an existing program using alter, delete, insert, cursor and page keys.**

CNC Operator Panel


CRT Display
Cursor Keys
Reset Key
Data Input Keys
Program/Edit Keys
Software Panel
Input Key
Start/Output Key
Function Keys
Page Change Keys
Soft Keys
Feed Rate Override.
Percentage Spindle Load Meter.
Power On.
Spindle Override
Cycle Start.
Feed Hold.
Fault Reset.
Emergency Stop.
Manual Pulse Generator.
GE Fanuc OM
CURSOR
PAGE
RESET
ALT
INS
SEL
CAN
POS
PRG
X
Y
Z
B
Z AT ATC
MAN ATC
TOOL NO
50
100
0
150
%
KW %
Bridgeport
MACHINING CENTRE

CNC Operator Panel Usage

Identify the location of the following elements on the control panel:

1. Mode Select - OPR/Alarm Function Key
    - A] Edit
    - B] Auto (Memory)
    - C] M.D.i.
    - D] Handle
    - E] Jog
    - F] Zero Return (ZRN)

2. Reset
    - A] Clears Alarm
    - B] Back to Start-Up Condition

3. Override Knobs
    - A] Spindle Override
    - B] Feed Rate Override
    - *C] Rapid Override
      *May have to set dry run to "ON" if no button exists.
      Will control rapid and feed.

4. Feed Hold
    - A] Stops axes motion
    - B] Coolant and spindle not stopped

CNC Operator Panel Usage

5. Manual Pulse Generator (Job Knob)
    - A] Increments Switchable
        - X1 = ____________________
        - X10 = ____________________
        - X100 = ____________________

6. Program/Edit Keys
    - A] Edit Mode
    - B] INS = __________________ of block
    - C] DEL = __________________of word
    - D] ALT = __________________ of word
    - E] EOB = __________________command
    - F] CAN = _______________ of last entry

7. Input Key Use
    - A] Tool Length Offsets
        1. D Word ________________
        2. H Word ________________
    - B] Work Coordinate Offsets
        1. G54-G59
            - a] 01 = _______________
            - b] 02 = _______________
            - c] 03 = _______________

CNC Operator Panel Usage

8. Power On
    - A] Enables drive motors

9. Emergency Stop
    - A] Panic button

10. Percentage Spindle Load Meter
    - A] Working percent of spindle

11. Fault Reset
    - A] Machine reset after error

12. Soft Keys
    - A] Left arrow (function menu)
    - B] Right arrow (operation menu)

CNC Operator Panel Usage

13. X Y Z B Keys
    - A] Use in zero return

14. Data input keys
    - A] Program entry or edit

15. Start/Output Key
    - A] Upload program to computer

16. Cursor Keys (Up and Down)
    - A] Search use

17. Page (Up and Down)
    - A] Different screens
    - B] Program editing

CNC Operator Panel Usage

18. Z AT
    ATC
    - A] Z axis at auto tool change position

19. Tool No.
    - A] Tells; when pushed, which tool no. is in spindle

CNC Operator Panel

1. M.D.i. the machine to start spindle in a clockwise direction at 400 RPM.
2. Turn coolant on
3. Use MPG to mill edge of part

Steps:

1. ______________________________
2. ______________________________
3. ______________________________
4. ______________________________
5. ______________________________
6. ______________________________
7. ______________________________

CNC Operator Panel

1. Correctly input a CNC program to drill one hole using keyboard.

Steps:

1. ______________________________

2. ______________________________

3. ______________________________

4. ______________________________

5. ______________________________

6. ______________________________

7. ______________________________

8. ______________________________

9. ______________________________

10. ______________________________

CNC Operator Panel

1. Correctly input a CNC program to mill a slot using the keyboard.

Steps:

1. ______________________________

2. ______________________________

3. ______________________________

4. ______________________________

5. ______________________________

6. ______________________________

7. ______________________________

8. ______________________________

9. ______________________________

10. ______________________________

CNC Operator Panel

1. Correctly edit a CNC program using EDIT keys found on the keyboard.

Program #______________________________

Steps:

1. ______________________________
2. ______________________________
3. ______________________________
4. ______________________________
5. ______________________________
6. ______________________________
7. ______________________________
8. ______________________________
9. ______________________________
10. ______________________________

Your Notes:

Your Notes:

Section Five

Your Notes:

Session Five

During this two hour segment, participants will:

1. **Discuss the importance of Alarm Code Numbers as found on FANUC and similar controls.**
2. **Discuss Alarm contents.**
3. **Define and explain important letters and numbers pertaining to Alarm Codes.**
4. **Explain what to do when clear Alarm message are on the CRT screen.**

Alarm List

Trouble shooting by Alarm Display...

Alarm List

When an alarm occurs, the alarm message is automatically displayed on the CRT.

Classification	*Alarm Number*
Program errors, operation errors	000-222

The next few pages give the alarm number and the contents of that alarm.

Alarm Codes

Number	Contents
000	A parameter which requires the power off was input, turn off power.
001	TH alarm (A character with incorrect parity was input). Correct the tape.
002	TV alarm (The number of characters in a block is odd). This alarm will be generated only when the TV check is effective. Correct the tape.
003	Data exceeding the maximum allowable number of digits was input. (Refer to the item of max. programmable dimensions.)
004	A numeral or the sign " - " was input without an address at the beginning of a block.
005	The address was not followed by the appropriate data but was followed by another address or EOB code.
006	Sign " - " input error (Sign " - " was input after an address with which it cannot be used. Or two or more " - " signs were input.)
007	Decimal point " . " input error (A decimal point was input after an address with which it can not be used. Or two decimal points were input.)
009	Unusable character was input in significant area.
010	An unusable G code was commanded.
011	Feedrate was not commanded to a cutting feed or the feedrate was inadequate.

(Note) "T" is a general term for 0-TC and 00-TC.
"M" is a general term for 0-MC and 00-MC.

Number	Contents
014	In variable lead threading, the lead incremental and decremental outputted by address K exceed the maximum command value or a command such that the lead becomes a negative value is given. T only
	A synchronous feed is specified without the option for threading / synchronous feed. M only
015	The number of the commanded axes exceeded that of simultaneously controlled axes. M only
021	An axis not included in the selected plane (by using G17, G18, G19) was commanded in circular interpolation.
023	In circular interpolation by radius designation, negative value was commanded for address R. M only
027	No axis is specified in G43 and G44 blocks for the tool length compensation type C. Offset is not canceled and another axis is offset for the tool length compensation type C.
028	In the plane selection command, two or more axes in the same direction are commanded.
029	The offset valus specified by H code is too large. M only
	The offset valus specified by T code is too large. T only
030	The offset number specified by H code for tool length offset or cutter compensation is too large. M only
	The offset number in T function specified for tool offset is too large. T only
031	In setting an offset amount by G10, the offset number following address P was excessive or it was not specified.
032	In setting an offset amount by G10, the offset amount was excessive.

Alarm Codes

Number	Contents
033	A point of intersection cannot be determined for cutter compensation C. M only
	A point of intersection cannot be determined for tool nose radius compensation. T only
034	The start up or cancel was going to be performed in the G02 or G03 mode in cutter compensation C. M only
	The start up or cancel was going to be performed in the G02 or G03 mode in tool nose radius compensation. T only
035	G39 is commanded in cutter compensation B cancel mode or on the plane other than offset plane. M only
	Skip cutting (G31) was specified in tool nose radius comapensation mode. T only
036	Skip cutting (G31) was specified in cutter compensation mode. M only
037	G40 is commanded at the plane other than offset plane in cutter compensation B. The plane selected by using G17, G18 or G19 is changed in cutter compensation C mode. M only The offset plane is switched in tool nose radius compensation. T only
038	Overcutting will occur in cutter compensation C because the arc start point or end point coincides with the arc center. M only
	Overcutting will occur in tool nose radius compensation because the arc start point or end point coincides with the arc center. T only
039	Chamfering or corner R was specified with a start-up, a cancel, or switching between G41 and G42 in tool nose radius compensation. The program may cause overcutting to occur in chamfering or corner R. T only
040	Overcutting will occur in tool nose radius compensation in a canned cycle G90 or G94.T only

Number	Contents
041	Overcutting will occur in cutter compensation C. M only
	Overcutting will occur in tool radius compensation. T only
042	Tool position compensation is commanded in tool radius compensation. M only
044	One of G27 to G30 is commanded in canned cycle mode. M only
046	Other than P2, P3 and P4 are commanded for 2nd, 3rd and 4th reference point return command.
050	Chamfering and corner R are commanded in the thread cutting block.
051	Improper movement or the move distance of the block next to that for which chamfering and corner R are commanded.
052	The block next to the block for which chamfering and corner R are commanded is not G01.
053	9n the chamfering and corner R commands, two or more of I, K and R are specified. Otherwise, the character after a comma (",") is not C or R in direct drawing dimensions programming.
054	A block in which the chamfering or the corner R was specified includes a taper command.
055	In the block for which chamfering and corner R are commanded, the move distance is commanded less than the corner R amount.
056	Neither the end point nor angle is specified in the command for the block next to that for which only the angle is specified (A). In the chamfering command, I (K) is commanded for the X (Z) axis.
057	Block end point is not calculated correctly in direct dimension drawing programming.
058	Block end point is not found in direct dimension drawing programming.
059	The program with the selected number cannot be searched, in external program number search.

Alarm Codes

Number	Contents
060	Commanded sequence number was not found in the sequence number search.
061	Address P or Q is not specified in G70, G71, G72, or G73 command. T only
062	• The depth of cut in G71 or G72 is zero or negative value. • The repetitive count in G73 is zero or negative value. • The negative value is specified to Δi or Δk in G74 or G75. • A value other than zero is specified to address U or W, though Δi or Δk is zero in G74 or G75. • A negative value is specified to Δd, though the relief direction in G74 or G75 is determined. • Zero or a negative value is specified to the height of thread or depth of cut of 1st time in G76. • The specified minimum depth of cut in G76 is greater than the height of thread. • An unusable angle of tool tip is specified in G76.
063	The sequence number specified by address P in G70, G71, G72, or G73 command cannot be searched. T only
065	◦ G00 or G01 is not commanded at the block with the sequence number which is specified by address P in G71, G72, or G73 command. Tonly ◦ Address Z (W) or X (U) was commanded in the block with a sequence number which is specified by address P in G71 or G72, respectively.
066	An unallowable G code was commanded between two blocks specified by address P and Q in G71, G72 or G73. T only
067	G70, G71, G72, or G73 command with address P and Q was specified in MDI mode. T only
069	The final move command in the blocks specified by P and Q of G70, G71, G72 and G73 ended with chamfering or corner R. T only

Number	Contents
070	The memory area is insufficient.
071	The address to be searched was not found. Or the program with specified program number was not found in program number search.
072	The number of programs to be stored exceeded 63 or 125 (option).
073	The commanded program number has already been used.
074	The program number is other than 1 to 9999.
076	Address P was not commanded in the block which includes an M98 command or a G65 command.
077	The subprogram was called in three or five folds.
078	A program number or a sequence number which was specified by address P in the block which includes an M98, M99, M65 or G66 was not found.
079	The contents of the program stored in the memory did not agree with that in tape in collation.
080	In the area specified by parameter ε, the measuring position reach signal does not come on. (Automatic tool compensation function) T only
081	Automatic tool compensation was specified without a T code. (Automatic tool compensation function) T only
082	T code and automatic tool compensation were specified in the same block. (Automatic tool compensation function) T only
083	In automatic tool compensation, an invalid axis was specified or the command is incremental. (Automatic tool compensation function) T only
085	When entering data in the memory by using ASR or Reader / Puncher interface, an overrun, parity or framing error was generated. The number of bits of input data or setting of baud rate is incorrect.

Alarm Codes

Number	Contents
086	When entering data in the memory by using Reader / Puncher interface, the ready signal (DR) of reader / puncher was turned off.
087	When entering data in the memory by using Reader / Puncher interfece, though the read terminate command is specified, input is not interrupted after 10 characters read.
090	The reference point return cannot be performed normally because the reference point return start point is too close to the reference point or the speed is too slow.
092	The commanded axis by G27 (Reference point return check) did not return to the reference point.
094	P type cannot be specified when the program is restarted. (After program interruption, the coordinate system setting operation was performed.)
095	P type cannot be specified when the program is restarted. (After program interruption, the external work offset amount changed.)
096	P type cannot be specified when the program is restarted. (After program interruption, the work offset amount changed.)
097	P type cannot be directed when the program is restarted. (After power ON, after emergency stop or P / S 94 to 97 reset, no automatic operation is performed.)
098	A command of the program restart was specified without the reference point return operation after power ON and emergency stop, and G28 was found during search.
099	After completion of search in program re - start, a move command is given with MDI.
100	Setting data PWE is set to 1. Turn it to 0 and reset the system.
101	The power was turned off while rewriting the contents of the memory in the part program storage & editing operation. When this alarm is generated, set the setting data PWE to I and turn on the power while pushing the DELET to clear the memory.
110	The absolute value of fixed decimal point display data exceeds the allowable range.
111	The calculation result of macro instruction exceeds the allowable range (-2^{32} to $2^{32}-1$).
112	Division by zero was specified. (including tan 90°)
113	A function which cannot be used in custom macro is commanded.
114	An undefined H code is designated in G65 block. For custom macro A
	There is an error in other formats than <Formula>. For custom macro B
115	A value not defined as a variable number is designated.
	The header contents are improper. This alarm is given in the following cases: High speed cycle machining 1. The header corresponding to the specified call machining cycle number is not found. 2. The cycle connection data value is out of the allowable range (0 - 999). 3. The number of data in the header is out of the allowable range (1 - 32767). 4. The storing start data variable number of executable format data is out of the allowable range (#20000 - #85535). 5. The last storing data variable number of executable format data is out of the allowable range (#85535). 6. The storing start data variable number of executable format data is overlapped with the variable number used in the header.
116	The variable number designated with P is forbidden for assignment.
	The left side of substitution statement is a variable whose substitution is inhibited.
118	The nesting of bracket exceeds the upper limit (quintuple).
119	The argument of SQRT or BCD is negative.
	The SQRT argument is negative. Or BCD argument is negative, and other values than 0 to 9 are present on each line of BIN argument. For custom macro B
122	The macro modal call is specified in double. M only

Alarm Codes

Number	Contents
123	Macro control command is used during DNC operation.
124	DO - END does not correspond to 1 : 1.
125	The unusuable address is used in G65 block. For custom macro A < Formula > format is erroneous. For custom macro B.
126	In DOn, 1 ≤ n ≤ 3 is not established.
127	NC and macro commands are confused.
128	The sequence number specified in the branch command was not 0 to 9999. Or, it cannot be searched.
129	An address which is not allowed in < Argument Designation > is used. T only
130	In axis control, a 3rd axis control command was given by PMC during Cf control. On the contrary, an attempt was made for Cf control from PMC during axis control.
131	Five or more alarms have generated in external alarm message.
132	No alarm No. concerned exists in external alarm message clear.
133	Small section data is erroneous in external alarm message or external operator message.
135	Without any spindle orientation, an attempt was made for spindle indexing. T only
136	A move command of other axes was specified to the same block as spindle indexing addresses C, H. T only
137	A move command of other axes was specified to the same block as M - code related to spindle indexing. T only
139	An axis is selected in commanding by PMC axis control.
141	G51 (Scaling ON) is commanded in the tool offset mode. M only
142	Scaling magnification is commanded in other than 1 - 999999. M only

Number	Contents
143	The scaling results, move distance, coordinate value and circular radius exceed the maximum command value. M only
144	The coordinate rotation plane and arc or tool offset C plane differ from each other. M only
145	The condition at the polar coordinate interpolation start or cancel is not correct. · G112/G113 are commanded by other mode than G40. · There are error at the plane selection. (Error of parameter setting) T only
146	G code which cannot be commanded are specified during the polar coordinate interpolation mode. T only
148	Automatic corner override deceleration rate is out of the settable range of judgement angle. Check parameter No. 0213, 0214, 0215. M only
150	Tool Group No. exceeds the maximum allowable value. M only
151	The tool group commanded in the machining program is not set. M only
152	The number of tools within one group exceeds the maximum value registerable. M only
153	A T-code is not stored in the due block. M only
154	When the group is not commanded, H99 or D99 was commanded. M only
155	In the machining program, M06 and T code in the same block do not correspond to the group in use. M only
156	P and L commands are missing at the head of program in which the tool group is set. M only
157	The number of tool groups to be set exceeds the maximum allowable value.

Alarm Codes

Number	Contents
158	The tool life to be set is too excessive. M only
159	During setting program execution, power was OFF. M only
160	Different M code is commanded in Heads 1 and 2 as waiting M code. 0TT only
165	An attempt was made to execute a program of an even number in Head 1 or an odd program number in Head 2. 0TT only
175	Conditions when performing circular interpolation start or cancel not correct. 1. G107 is not commanded simultaneously with rotation axis radius. 2. G107 is commanded simultaneously with two axes. 3. G107 is commanded during cutter radius R compensation. T only.
176	G code is commanded during circular interpolation mode when commands cannot be performed. T only
178	Commanded in the G41/G42 mode.
179	The number of controlled axes set by the parameter 597 exceeds the maximum number.
190	In the constant surface speed control, the axis specification is wrong. (Misprogram) M only
197	The program commanded the CI axis to move when the COFF signal was ON.
200	In the rigid tap, an S value is out of the range or is not specified. (programming error)
201	In the rigid tap, no F value is specified. (programming error)
202	In the rigid tap, spindle distribution value is too large (system error)
203	In the rigid tap, position for M29 or an S command is incorrect.

Number	Contents
204	In the rigid tap, an axis movement is specified between M29 and G84 (G74) blocks. (programming error)
205	When G84 (G74) block is executed.
210	M198 and M099 are executed in the schedule operation. M198 is executed in the DNC operation.
211	G31 is commanded in the per revolution command when the high-speed skip option is provided.
212	The direct drawing dimensions programming is commanded for the plane including an additional axis. M only The direct drawing dimensions programming is commanded for the plane other than the Z-X plane. T only
213	Movement is commanded for the axis to be synchronously controlled. T only
214	Coordinate system is set or tool compensation of the shift type is executed in the synchronous control. T only
217	G251 is further commanded in the G251 mode. T only
218	P or Q is not commanded in the G251 block, or the command value is out of the range. T only
219	G251 and G250 are not independent blocks. T only
220	In the synchronous operation, movement is commanded by the NC program or PMC axis control interface for the synchronous axis. T only
221	Polygon machining synchronous operation and axis control or balance cutting are executed at a time. T only
222	Input and output are executed at a time in the background edition. M only

Alarm Code Worksheet

I. Define

A] M only ____________________

B] T only ____________________

II. Explain

A] Code 065 ____________________

B] Code 074 ____________________

C] Code 010 ____________________

88

Alarm Code Worksheet

D] Code 030

E] Code 023

F] Code 072

Alarm Code Worksheet

III. Apply your knowledge

A code number comes on your CRT screen that is larger in value than #222.

Does the alarm have anything to do with program or operation errors?

______ Yes
______ No

Explain your answer:

Your Notes:

Your Notes:

Sections Six & Seven

Your Notes:

Training Daily Guide

Sessions Six and Seven

During this four hour segment, participants will:

1. **Define "Tool Length Offset"
Referring to this manual.**
2. **Explain "G" Codes: G43, G44 and G49 which deal with tool length compensation.**
3. **Understand command format using G43 or G44 in the CNC program.**
4. **Apply "H" and "D" words used for tool offsets.**
5. **Understand preset tool methodology.**
6. **Review tool presetter information pages.**
7. **Complete a tool length setting using a tool presetter on the shop floor.**
8. **Complete setup sheet - with tool offset valve.**

Tool Offsets

One tool length offset is known, a "G" Code will cap up the offset length for that tool in the program.

Command format:

N __________ G43 Z__________ H __________

Where:
N = Sequence number
G43 = Tool length offset positive direction
Z = Z coordinate of tool
H = Length amount
Ex. = 5.250 inches

N __________ G44 Z__________ H __________

Where:
N = Sequence number
G43 = Tool length offset negative direction
Z = Z coordinate of tool
H = Length amount
Ex. = –5.250 inches

Tool cancel
G49 = Cancel offset amount

Tool Offsets

Two words which apply to tool offsets.

"H" Word	Used generally for tool length values
"D" Word	Used generally for tool radius values

Tool offset numbers in control

01 - 21	[Use for "H" words]
22 - 64	[Use for "D" words]

Tool Offset

Example: Where found in program

```
:0 2021

N5  G20  G40  G49  G80  G90

N10  T2  M6  (.T Dia End Mill)

N15  51600 M3

N20  60 G54 X0 Y6.

N25  G43  Z.1  H2  M8

N30  G1  Z–.1  H2  M8
```

G43 ______________________________

Z.1 ______________________________

H2 ______________________________

M8 ______________________________

Tool Presetter

Preset Tool Method

Your Notes ____________________

Tool Presetter

Overview of machine

OVERVIEW

The Parset MASTER control is based on the principle that there are OPERATIONS that are done to COMPONENTS.

The four basic operations available are:

STORE / ADD
MEASURE / PRESET
MODIFY / DELETE / CANCEL
VIEW / DISPLAY

The three basic components that the operations are applied to are:

ZERO GAGE
TOOL / TOOL ASSEMBLY
TOOL PROGRAM

By combining an operation with a component, you store information into the control and recall that information when necessary to measure or preset a particular tool.

COMPONENTS

ZERO GAGE: This is the calibration point which is used in the control to calculate the measurements. On all controls, you must the crosshairs of the projector on the Zero Gage tip, before you STORE / ADD the zero gage. This is necessary because the glass scales are read to get the current position of the machine when you STORE / ADD a zero gage.

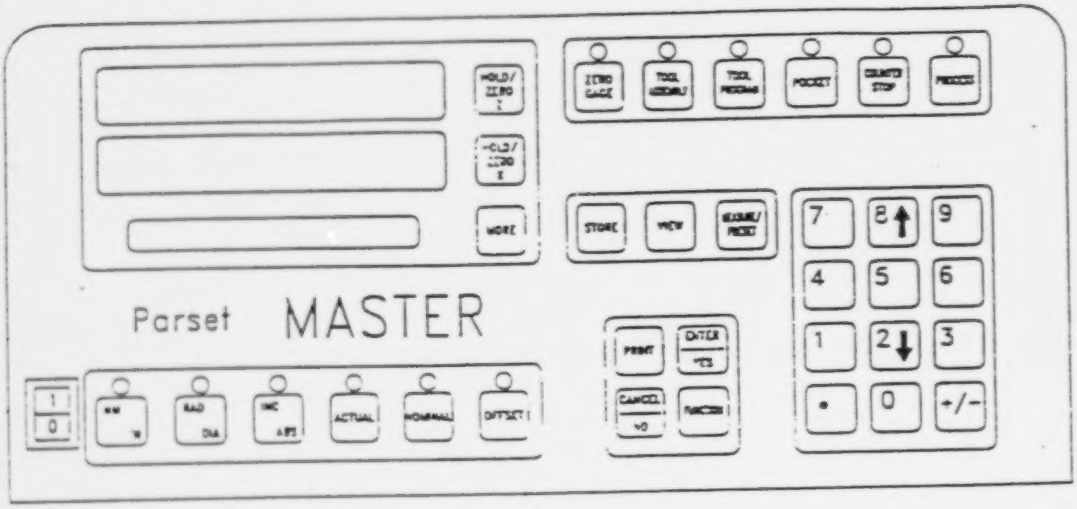
The data that is stored with a Zero Gage is :

ZERO GAGE #	- ID number
X value	- X value of the calibration point
Z value	- Z value of the calibration point

TOOL / TOOL ASSEMBLY. A TOOL is a holder, cutting tool, retention knob, etc. that is ready to be placed in the machine tool. The only difference between a TOOL and TOOL ASSEMBLY is that a TOOL ASSEMBLY is stored in the memory of the control, and a TOOL is not.

PARSET MASTER

The Parset Master is a DRO type control designed exclusively for use on tool measuring machines.


HOLD/
ZERO
Z
MORE
ZERO
GAGE
TOOL
TOOL
POCKET
COUNTER
STOP
STORE
VIEW
7
8
9
4
5
6
1
2
3
•
0
+/-
Parset MASTER
PRINT
ENTER
CANCEL
1
0
RAD
DIA
INC
ABS
ACTUAL
NOMINAL
OFFSET

STARTUP

When the control is first turned on, it will go through a self diagnostic procedure that takes about ten seconds. When it is finished, the control will beep, and the prompt display will show "Reference scales". To reference the scales, move the joystick in the X axis until the control beeps twice in rapid succession. Then move the Z axis until the control beeps twice in rapid succession. If you want, you can move both axes at the same time. Once the control has beeped for both axes, the control is referenced.

BEST COPY AVAILABLE

5) Type in the Z value for this tool from the number pad. This is normally the nominal length of the tool being stored.

6) Press ENTER - Prompt display shows "X:?"

7) Type in the X value for this tool from the number pad. This is normally the nominal diameter of the tool being stored.

8) Press ENTER - prompt display shows "ZG #:?"

9) Type in the zero gage number for this tool from the number pad. This must have been already stored with Store Zero Gage.

10) Press ENTER - Prompt display shows "TA stored" for a couple of seconds.

STORING A TOOL PROGRAM

1) Press STORE - prompt display shows "Store what?"

2) Press TOOL PROGRAM - prompt display shows "TP #:?"

3) Type in the tool program ID from the number pad. The ID can be up to 10 digits long, and hypens (-) and decimal points (.) are allowed.

4) Press ENTER - prompt displays shows "Step 1" for a couple of seconds.

5) Prompt displays "TA or ZG".

6) Press either TOOL ASSEMBLY or ZERO GAGE depending on what the first step of the program is going to be.

IF YOU CHOOSE TA

7) Press TOOL ASSEMBLY - prompt displays shows "TA #:?"

8) Type in the tool assembly ID using the number pad. The tool assembly must have already been stored using Store Tool Assembly.

9) Press ENTER - Prompt display shows "Pkt #:?"

10) Type in the pocket number for this step of the program using the number pad.

11) Press ENTER - prompt display shows "TA step stored" for a couple of seconds.

length and diameter of a given tool. The only information you need to be able to Measure/Preset is the ID of the component.

MEASURING A TOOL

1) Put the tool in the Parsetter spindle

2) Press MEASURE/PRESET - prompt display shows "Measure what?"

3) Press ZERO GAGE - prompt display shows "ZG #:?"

4) Type in the Zero Gage number from the number pad.

5) Press ENTER

6) You can now measure the tool.

MEASURING A TOOL ASSEMBLY

1) Put the tool in the spindle of the Parsetter.

2) Press MEASURE/PRESET - Prompt display shows "Measure what?"

3) Press TOOL ASSEMBLY - prompt display shows "TA #:?"

4) Type in the tool assembly ID number from the number pad

5) Press ENTER

6) You can now measure the Tool Assembly

VIEW / DISPLAY: This allows you to look at what components are already stored in the memory of the control. You would use this if you needed to see if the component you were looking for was in the control or if someone had deleted it.

VIEWING A ZERO GAGE

1) Press VIEW - prompt display shows "View what?"

2) Press ZERO GAGE

3) Use the UP ARROW and DOWN ARROW keys on the number pad to scroll the list of ZERO GAGES. The numbers that you see in the Z and X displays are the values of the Zero gage.

4) To finish the view operation, either scroll off the end of the list, or press the CANCEL / NO key.

VIEWING A TOOL ASSEMBLY

1) Press VIEW - prompt display shows "View what?"

2) Press TOOL ASSEMBLY

3) Use the UP ARROW and DOWN ARROW keys on the number pad to scroll the list of tool assemblies. The numbers that you see in the Z and X displays are the nominal values of that tool. To see which Zero Gage the tool assembly is using, press the MORE key once. This will change the prompt display so it will show the Zero gage. To change back to viewing the Tool Assemblies, press the MORE key once again.

4) To finish the view operation, either scroll off the end of the list, or press the CANCEL / NO key.

MODIFY / DELETE / CANCEL: This function allows you to permanently remove a given component from the memory of the control. This would be done either because that component was no longer used, or the memory of the control is full and you need to free some of it.

DELETING A ZERO GAGE

WARNING - DO NOT DELETE ZERO GAGE # 0. This Zero Gage in required for the control to function correctly.

1) Press CANCEL / NO - prompt display shows "Clear what?"

2) Press ZERO GAGE - prompt display snows "ZG #:?"

3) Using the number pad, type in the number of the Zero Gage that you wish to delete.

4) Press ENTER

5) If the prompt display shows "ZG Cleared", then that zero gage was removed from the memory of the control.

6) If the prompt display shows "ZG not cleared" - "Used in TA or TP", it means that either a Tool assembly or a Tool program is using that Zero gage, and it cannot be cleared.

DELETING A TOOL ASSEMBLY

1) Press CANCEL / NO - prompt display shows "Clear what?"

2) Press TOOL ASSEMBLY - prompt display shows "TA #:?"

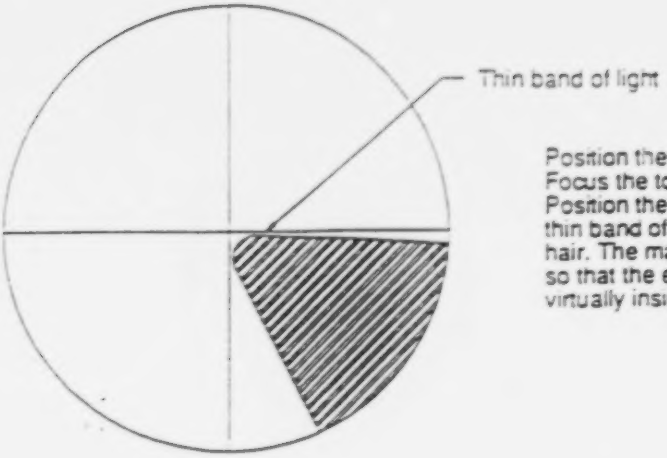
3) Using the number pad, type in the number of the Tool Assembly that you wish to delete.

4) Press ENTER

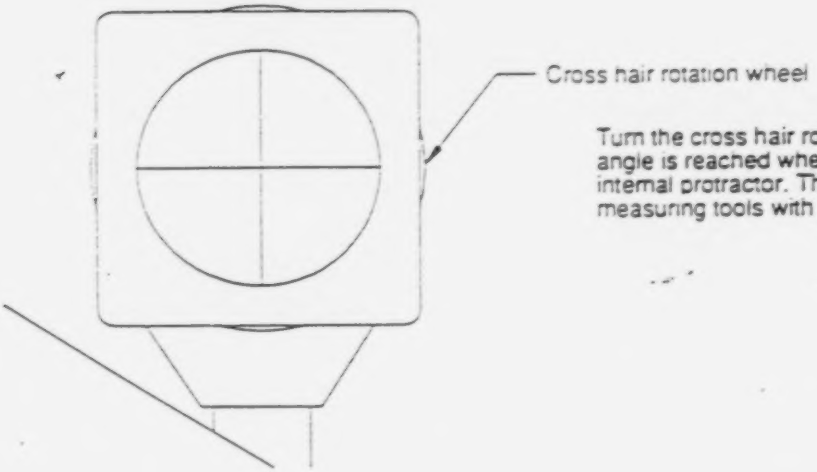
5) If the prompt display shows "TA Cleared", then that Tool Assembly was removed from the memory of the control.

6) If the prompt display shows "TA not cleared" - "Used in TP", it means that a Tool program is using that Tool Assembly, and it cannot be cleared.

Projector Operation



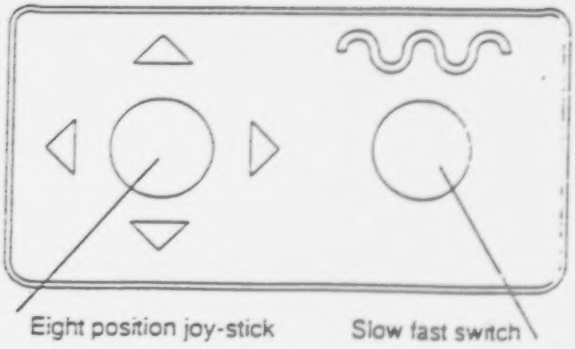
Position the tool tip in the projectors field of vision. Focus the tool by rotating until the image is sharp. Position the tool tip as sown in Figure , leaving a thin band of light between the tool tip and the cross hair. The magnification of the projector is 20 times so that the error caused by this thin band is virtually insignificant.



Turn the cross hair rotation wheel until the desired angle is reached when viewed in the projectors internal protractor. This feature is used when measuring tools with leading edges.

The Parsetter 240 is equipped with a lamp dimmer circuit to extend bulb life. If the machine is idle for 20 minutes the power to the projector bulb is interrupted which dims the bulb. Touching the joy-stick turns this back on.

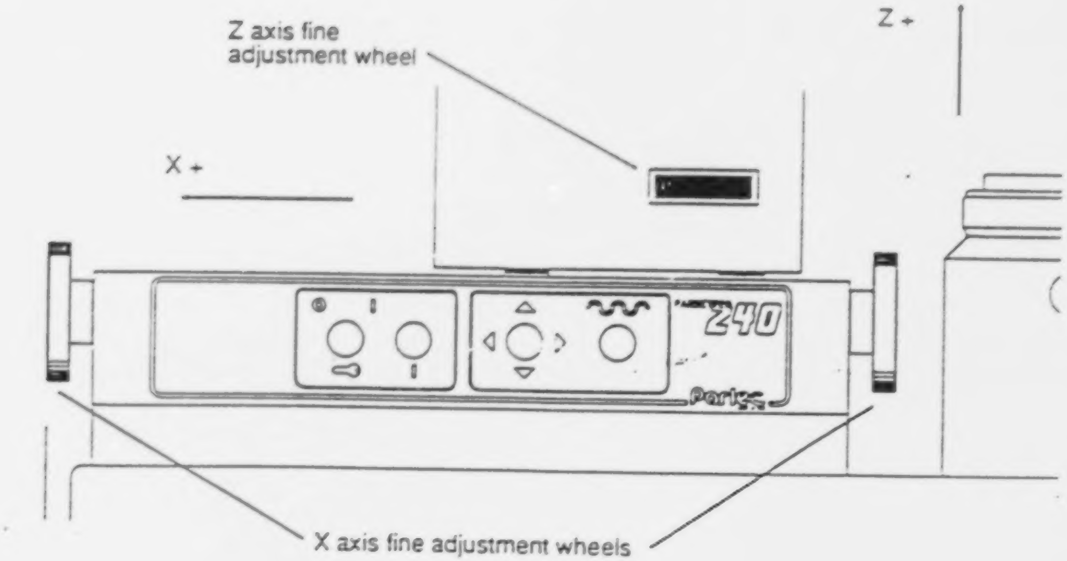
Operations Guide: Axis Control


Eight position joy-stick
Slow fast switch

Rough positioning is accomplished by moving the joy-stick in the direction desired. Two speeds are available for this jog feature, fast and slow. These are selected by depressing or releasing the slow fast switch on the control panel. Fine adjustment is made utilizing the thumb wheels.

The slow fast switch is a dead man switch which defaults to slow speed. Pressing and holding in the switch engages high speed.

Pushing the joy-stick in the direction of an arrow will move the projector in that direction. Pushing the joy stick at 45° between two arrows will move the projector in both directions.


Z axis fine adjustment wheel
Z +
X +
240
Parise
X axis fine adjustment wheels

Fine adjustment range is unlimited. Two thumb wheels are provided for the X axis. Arrows on the machine indicate which direction to turn the thumb wheels to get the desired movement.

Tool Offset Presetting Basics

What is presetting?

Presetting is the act of positioning a tool tip at a specific set of coordinates. These locations are called the nominal tool dimensions.

Why should you preset?

There are four basic reasons to preset. These are as follows:

1. Certain diametrally adjustable tools (boring tools, reamers) require a diameter setting to produce a good part.
2. Multiple spindle machines with the spindles at different heights require that tools be set for each spindle to produce identical parts.
3. Special requirements of the program for clearance or machining conditions.
4. Tools used on older controls that have no tool offset capabilities.

How to preset an adjustable tool.

1. Diameter adjustable tools.

Position the machine so that the diameter (or radius) desired is showing on the x axis display field of the control. Adjust the cutting bringing the tool tip to the cross hairs of the projector.

2. Length adjustable tools.

2.1. Positive lock end mill holders.

Position the machine so that the length desired is showing on the Z axis display field of the control. Adjust the tool length using the back up screw. Tighten the side lock screw. Final tighten the side lock screw and snug the back up screw in a tightening fixture, not the presetter spindle.

2.2 Collet chucks.

Collets draw down .005 - .015 as they are tightened. Position the machine so that the length desired + .015 is showing on the Z axis display field of the control. Tighten the collet nut. Reposition the machine to measure the actual tool length. Snug up the back up screw.

NOTE: Never tighten a collet chuck with the tool snugged up against a back up screw. This prevents the proper pull down action of the collet and can reduce the gripping power by as much as 50%.

Tool Measuring Basics

What is tool measuring?

Tool measuring is the act of establishing a tools actual length and diameter. This is different from presetting in that no adjustment to the tool length and diameter to achieve specific dimensions is made.

Why should you measure your tools?

You can't make a good part on a CNC machine until you at least know the tool length. At some point in the machine set up process you must establish this dimension either by a measuring operation or by touching off.

How to measure a tool.

1. Tools with small radii

Position the machine until the tool tip is in the field of vision of the projector. Focus the tool by rotating until the image is sharp. Clamp the tool. Position the machine until the tool tip is aligned with the cross hairs of the projector.

2. Tools with large radii

Position the machine and focus the tool as above. Position the machine and measure either the length or diameter. Press the hold button for the axis measured. Position machine and measure the other axis. Print a label if desired. Release the axis hold by depressing the button.

3. Drills

Drilled holes are generally dimensioned as shown in figure one. It is very easy to measure a drill to produce this feature by aligning the drills outside diameter and point with the projector cross hairs.

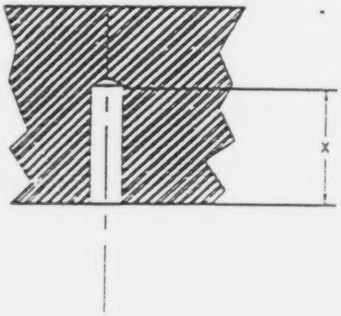


Figure 1. Dimension of drill depth.

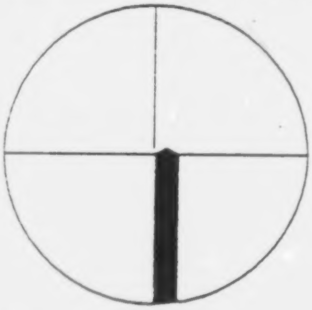


Figure 2. Measuring set length to full drill depth.

4. Runout

The runout of drills and other tools can be quickly and easily checked. Locate one edge of the drill on the cross hair of the projector. Press the absolute/incremental switch on the control so the each axis reads zero. Rotate the tool 180°. Any run will be easily seen and can be measured by moving projector so that the other edge is located on the cross hair. Point runout of drills can be measured in the same way.

Maintenance & Trouble Shooting

Replacing and centering the light bulb:

Turn off the power switch. Loosen the set screw and pull the lamp socket out. Pull the old bulb out. Using a soft cloth pick up the new bulb and insert into the socket. Never grip the bulb with bare fingers as this may adversely affect bulb life. Push the socket back in the housing making sure that the filament is in the vertical position. Adjust the socket in and out of the lens housing to achieve the greatest projector illumination. Tighten the set screw.

Preventive maintenance:

Standard spindles

Periodically oil spindle oiler with 30 weight oil.

Optional air vacuum spindles

Periodically check the water trap. If an excessive amount of water has accumulated, drain the reservoir.

Periodically check the oiler. if the oil level is low refill the reservoir.

Trouble shooting guide:

We turn the machine on and nothing happens! Make sure machine is plugged in.

Machine is plugged in and nothing happens! Check fuse and replace as required.

Projector light works but the control is not on! Turn control on with switch located in front panel.

When the tool pressure switch is engaged nothing happens! Make sure air line is properly connected, then increase pressure by turning the pressure control valve clockwise.

Projector light is not on! Replace projector bulb. Before replacing the bulb make sure that the machine is not in a dimmer cycle. If the machine is not used for 20 minutes the light bulb will automatically dim to improve bulb life. Moving the joy-stick turns the light back on.

Axes will not move. Remove shipping bolts.

Thank you again for purchasing a Parsetter 240. If you need additional information on accessories or service please call 1-800-PARSET1

Considerations for the Preset Area

Organizational ideas

Do I have to reorganize my shop?

No you don't have to reorganize your shop to benefit from your PARSETTER. What is required is some discipline and planning.

Considerations for the preset area

1. Locate an assembly/breakdown bench adjacent to the machine. Equip this with tightening fixtures and a vise. Use this bench to assemble tools , install retention knobs and to do any final tightening.

2. Establish some method of communicating the tooling requirements for a given program to the preset operator.

3. DO equip your PARSETTER with a label printer.

4. Have enough toolholders to set up the next job while the current job is running.

5. Provide some means of transporting set up tool assemblies to the machine without banging the tapers together.

6. Send any required perishable tools (inserts,drills,taps) to the machine along with the tool assemblies.

Set up sheets

One method of communicating tooling requirements is through the use of a set up sheet. A sample of one type of set up sheet is shown on the next page.

Practical Operation of Presetter

Steps:

1. ____

2. ____

3. ____

4. ____

5. ____

6. ____

7. ____

8. ____

9. ____

10. ____

Reading = Z ____________________

Tool Offset

Complete setup sheet for Tool #1, after performing practical tool preset.

DATE ____________________

CNC MILL TOOL SETUP SHEET

OP NO.	PART NAME	PART #	FIXTURE NO.	PROGRAMMER	L/O #

TOOL NO.	TOOL DESCRIPTION	SPINDLE ADAPTER	TOOL ADAPTER	SETUP (TLO) HEIGHT	COMMENTS

NOTES

____

____

Section Eight

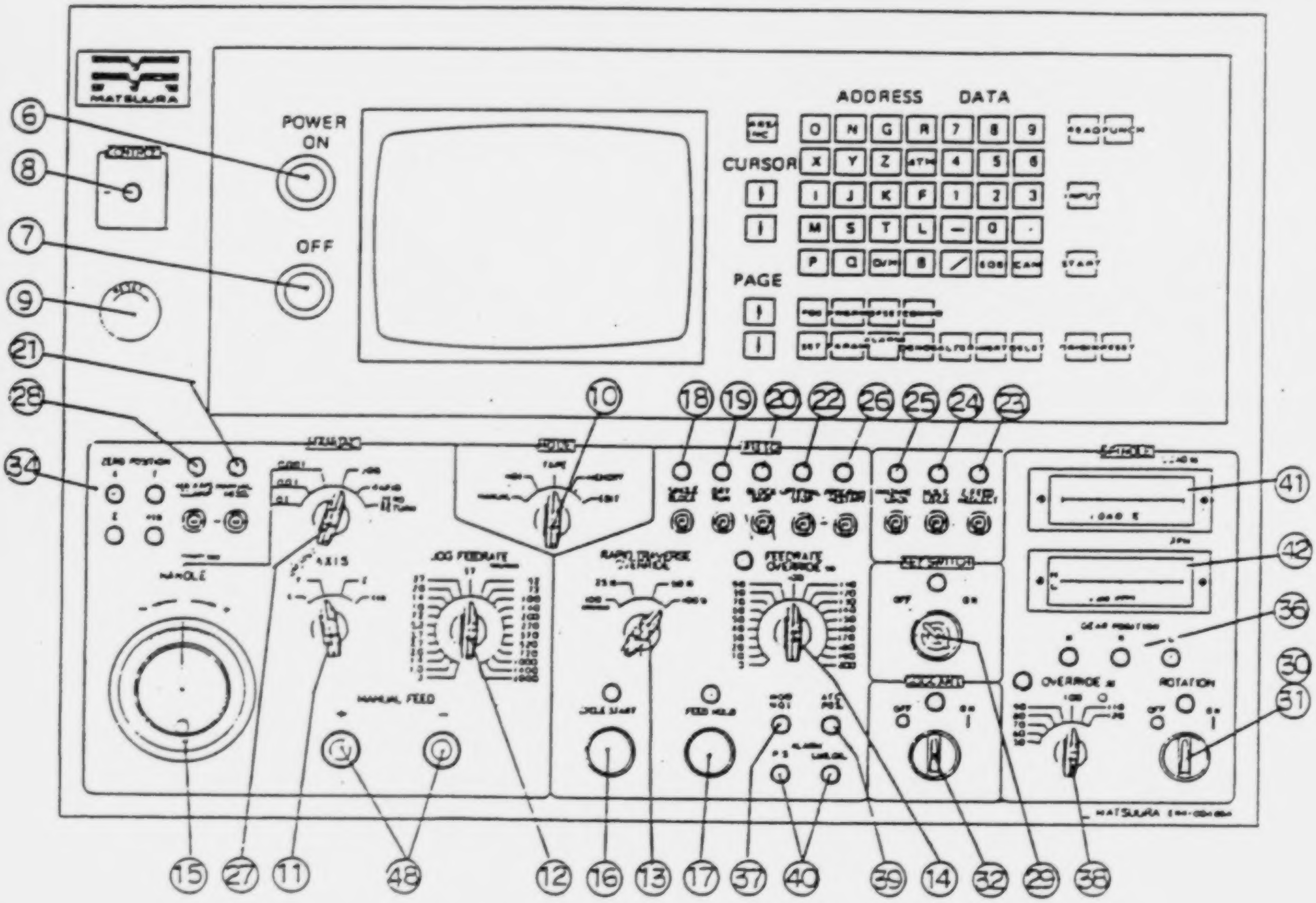
Your Notes:

Session Eight

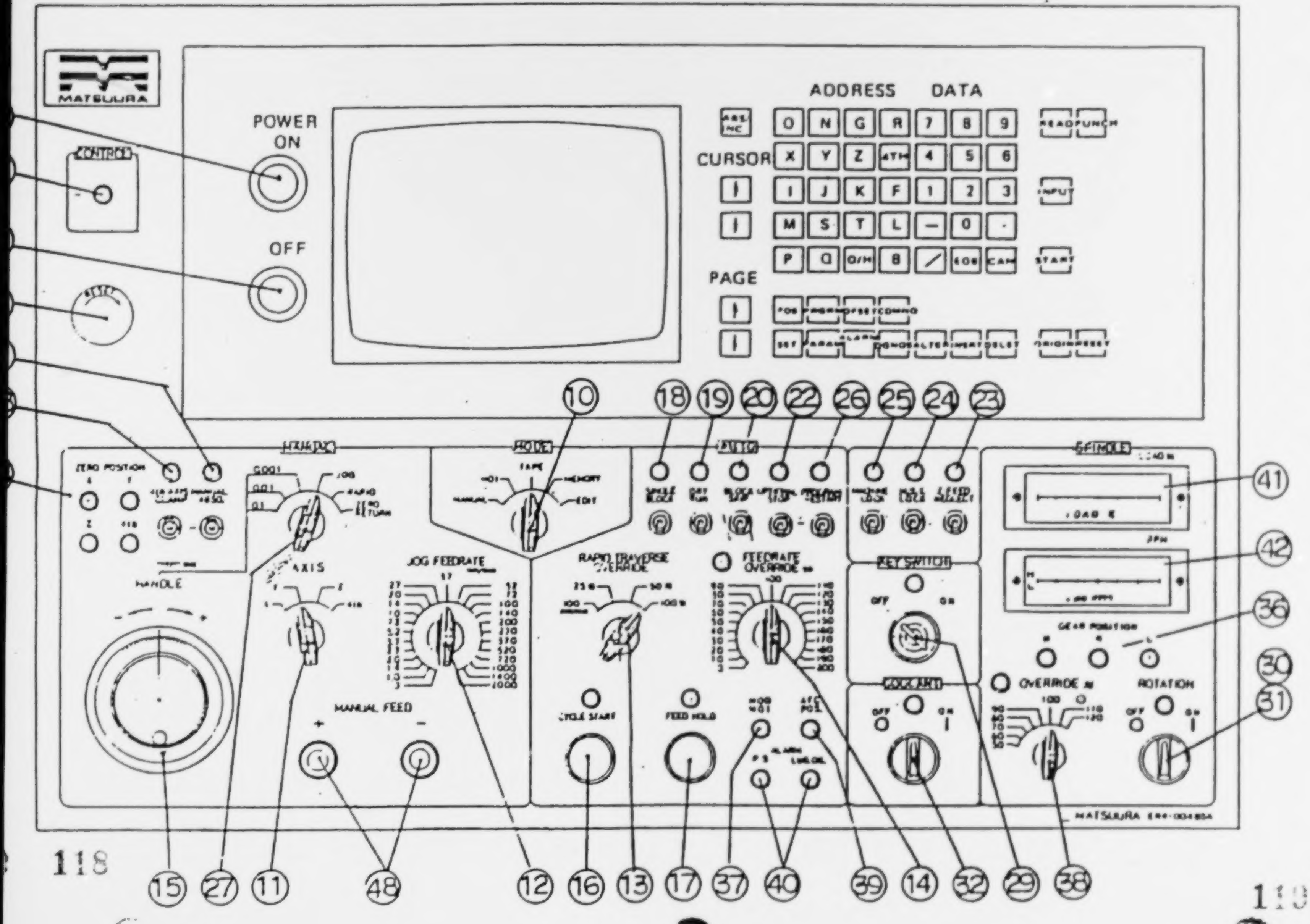
During this two hour segment, participants will:

1. Become familiar with the Matsuura CNC control panel.
2. Power up a CNC machine in the proper sequence of events.
3. Set the switches on the main operator's control panel to recommend settings.
4. Understand procedures regarding automatic operation of CNC machines.
5. Power down CNC machines in proper sequence of events.
6. Understand how to access the program directory (library) on the Matsuura CNC machine.

Matsuura Control Panel


⑥
⑧
⑦
⑨
㉑
㉘
㉞
⑩
⑱
⑲
⑳
㉒
㉖
㉕
㉔
㉓
㊶
㊷
㊱
㉚
(51)
⑮
㉗
⑪
㊽
⑫
⑯
⑬
⑰
㊲
㊵
㊴
⑭
㉜
㉙
㊳
MATSUURA
POWER ON
OFF
ADDRESS
DATA
CURSOR
PAGE
O
N
G
R
7
8
9
X
Y
Z
4
5
6
I
J
K
F
1
2
3
M
S
T
L
—
0
.
P
Q
8
/
EOB
CAN
READ
PUNCH
INPUT
START
AXIS
HANDLE
JOG FEEDRATE
MANUAL FEED
RAPID TRAVERSE OVERRIDE
FEEDRATE OVERRIDE
CYCLE START
FEED HOLD
OVERRIDE
ROTATION
GEAR POSITION
OFF
ON
MATSUURA

BEST COPY AVAILABLE


MATSUURA
POWER ON
OFF
CONTROL
RESET
ADDRESS DATA
ABS INC
CURSOR
PAGE
O N G R 7 8 9
X Y Z 4TH 4 5 6
I J K F 1 2 3
M S T L — 0 .
P Q O/H B / EOB CAN
READ PUNCH
INPUT
START
POS PRGRM OFSET COMND
SET PARAM ALARM DGNOS ALTER INSRT DELET
ORIGIN RESET
MANUAL
MODE
AUTO
SPINDLE
ZERO POSITION
X Y Z 4th
HANDLE
AXIS
JOG FEEDRATE
RAPID TRAVERSE OVERRIDE
FEEDRATE OVERRIDE %
KEY SWITCH
OFF ON
COOLANT
GEAR POSITION
OVERRIDE %
ROTATION
MANUAL FEED
+
−
CYCLE START
FEED HOLD
LOAD %
RPM
MATSUURA ER4-00480A
10
18
19
20
22
26
25
24
23
41
42
36
30
31
15
27
11
48
12
16
13
17
37
40
39
14
32
29
38

BEST COPY AVAILABLE

Matsuura Control Panel

Name the control features

6. ______________________________
7. ______________________________
8. ______________________________
9. ______________________________
10. ______________________________
11. ______________________________
12. ______________________________
13. ______________________________
14. ______________________________
15. ______________________________
16. ______________________________
17. ______________________________
18. ______________________________
19. ______________________________
20. ______________________________
21. ______________________________
22. ______________________________
23. ______________________________
24. ______________________________
25. ______________________________
26. ______________________________
27. ______________________________
28. ______________________________
29. ______________________________
30. ______________________________
31. ______________________________
32. ______________________________
33. ______________________________
34. ______________________________
35. ______________________________
36. ______________________________
37. ______________________________
38. ______________________________
39. ______________________________
40. ______________________________
41. ______________________________
42. ______________________________
43. ______________________________
44. ______________________________
45. ______________________________
46. ______________________________
47. ______________________________
48. ______________________________

Powering Up CNC Machines

Lubrication

1] Slide lube filled sufficiently
2] Coolant oil tank
3] Air lubricator

Spindle Taper

1] Make sure it is clean
2] Tapers of tool holders clean
3] Pull stud is securely tightened

Doors of NC Unit

1] Make sure door is securely closed

Turning Power On

1] Turn on main power switch and check if main power lamp is lit
2] Press power of button on operation panel *Note: some control require this to be pressed twice; first time for NC unit, second time for machine side*

Air Pressure

1] Air lubricator normally requires 70 psi

Manual Zero Return Axes*

1] Set mode switch (10) to zero return

2] Depress (+) button of the **Manual Feed**
Push button switch (48)

3] Continue pushing (+) push button unit all axes are at zero return position

4] The first zero return speed after power **ON** is set at 50% of the rapid feedrate.

5] Axes traverse if first performed at rapid feedrate, then when the axis runs over the deceleration DOG, it decelerates and stops at zero point. Zero position lamp is then lit.

**Note: some controls allow automatic zero return of all axes at power up.*

Automatic Operation (Memory Operation)

Automatic operation (memory operation)

Automatic operation is performed by the following procedures.

a) Make sure that the tools are arranged in the magazine according to the program data sheet.

b) Make sure that each axis is returned to each zero point manually after power ON.

c) Make sure that [PS ALARM] lamp and lamp(40) on the main control panel are extinguished.

d) Make sure that no alarm message is displayed on the CRT screen. Alarm message, if any, is displayed when the ALARM key on the CRT unit is depressed.

Automatic Operation (Memory Operation)

e) Make sure that a tool is not in the spindle.
In the presence of a tool, check the CRT screen to see whether the tool number is registered on the control. If it is not registered, set the tool number according to the parameters.

f) Check and make sure the tool length offset and cutter radius compensation values are input correctly.

g) Depress [FORWARD] switch for the chip conveyer (option).

h) Registration of program
Work programs made according to program FORMAT are registered into NC memory. Each program must have its own program number at the head of the program.

i) Make sure that each axis is positioned at the specific position shown by program data sheet and check the position display.

Automatic Operation (Memory Operation)

j) Set the switches on the main operation control panel.

MODE switch (10)	TAPE or MEMORY
RAPID TRAVERSE OVERRIDE switch (13)	Normally at 100%.
FEEDRATE OVERRIDE switch (14)	Normally at 100%.
SPINDLE OVERRIDE switch (38)	Normally at 100%.
SINGLE BLOCK switch (18)	ON or OFF
DRY RUN switch (19)	ON or OFF

(When DRY RUN switch is set to ON, set the JOG FEEDRATE switch (12) at the desired value.)

BLOCK SKIP switch (20)	ON or OFF
OPTIONAL STOP switch (22)	ON or OFF
Z-FEED NEGLECT switch (23)	ON or OFF
M,S,T LOCK switch (24)	ON or OFF
MACHINE LOCK switch (25)	ON or OFF
MIRROR IMAGE	ON or OFF
EDIT switch (29)	LOCK

NOTE: Underlined settings are normally used for machining operations.

k) The program is started by depressing the CYCLE START pushbutton (16) on the main operation control panel. The CYCLE START lamp is lit when the automatic operation is started.

Stopping the Machine During Operation

Stopping the machine during an automatic operation

The machine can be stopped during an automatic operation in various ways, as occasion requires.

1) Programming of M00 or M01 beforehand.

When M00 or M01 is commanded, spindle rotation, coolant and oil mist are stopped and the axis feed is temporarily stopped after completion of its block. (M01 is effective only when OPTIONAL STOP switch is ON.) Restart is effected by depressing CYCLE START pushbutton (16).

This function is used;
- to check the work dimensions.
- to exchange workpieces.
- to remove the cutting chips.
- to perform manual operation.
- to change the tools manually.

Temporary Stop by FEED HOLD

2) Temporary stop by FEED HOLD pushbutton (17)

Axis feed is decelerated and stopped by depressing FEED HOLD pushbutton switch during an auto. operation. When M.S.T. functions are already commanded, they are performed. The CYCLE START lamp (16) is extinguished and FEED HOLD lamp is lit.

FEED HOLD is ignored when the Z axis cutting feed is performed by a tapping cycle (G74 or G84).

FEED HOLD is effective on DWELL, the DWELL is stopped.

During a temporary stop by FEED HOLD, the following manual operations can be performed.

1. Manual axis feed.
   a) After a FEED HOLD stop, note down the coordinate values of the stopped points of each axis by means of the position display.
   b) Set MANUAL ABSOLUTE switch to ON or OFF.
   c) Perform the manual operation.
   d) Bring each axis back to the position of the noted coordinate value by using the position display.
   e) Change MODE select switch (10) back to TAPE or MEMORY.
   f) Depress the cycle start button (16) to re-start the auto. operation.

Manual Operation of the Machine

2. Manual operation of the machine
    - ON/OFF operation of the spindle rotation.
    - Rotation of magazine after spindle stop
    - Removing/inserting the tool holder to/from the spindle after spindle stop.

etc. can be performed.

Note: MDI operation cannot be performed during a temporary stop by FEED HOLD. To perform MDI operatin, the system must be reset once. If it is dessired to change the operation mode, wait until the operation is stopped with a single block.

Emergency Stop

5) Emergency Stop

When the machine must be stopped immediately during an operation, depress EMERGENCY STOP (9) button. Then, the power of the machine is shutt off, spindle rotation, axis feed are stopped immediately. Emergency stop is also applied to NC and servo system is turned off. All the motions of the machine are stopped.

To resume the operation, turn on POWER again, and return each axis to its zero return manually again.

Shutting off of the Power

a. Make sure that CYCLE START lamp (6) on the main operation panel is extinguished.

b. Make sure that each motion (axis feed or ATC motion etc.) of the machine is not performed. If the spindle is rotated, stop it.

c. Remove the tool if the tool is inserted into the spindle or in the ATC arm grip.

d. Clean the inside of spindle taper carefully, apply rust-preventive oil.

e. Return each axis (especially X axis) to the center to maintain the balance of the machine.

f. Depress [STOP] switch of the chip conveyer to stop the chip conveyer.

g. Depress EMERGENCY STOP pushbutton (9) on the main operation panel. Then the power of the machine is shut off and emergency stop is allied to the NC system, causing the servo system to be turned off.

h. Depress POWER OFF switch (7) on the CRT unit. Then the power of NC unit is shut off.

i. Turn the main power switch to OFF.

Matsuura CNC Machine

Procedure to bring up directory on CRT screen.

1] Place mode switch in **EDIT** mode.

2] Turn editor switch to **ON**.

3] Press **CANCEL** button.

4] Press **ORIGIN** button.

5] Press **CURSOR DOWN** button.

**Note: wait a few seconds and the programs residing in memory will be displayed on the machine.*

139

Matsuura CNC Control

Questions.

1. Reference switch No. 10. Which mode is selected to manually references the axes?

   ____

   ____

2. Which axis is reverenced first? Why?

   ____

   Why? ____

   ____

3. What tells you on the control that the axes are at their home or zero return position?

   ____

   ____

4. What does light No. 39 indicate?

   ____

   ____

Matsuura CNC Control

5. For automatic operation, what should the following switches be set to?

Switch	**On or Off**
Z Feed Neglect	____________________
M, S, T, Lock	____________________
Machine Lock	____________________
Mirror Image	____________________

6. Where should the axes X, Y, and Z position prior to shutting off the power? Why?

____

Why? ____

____

7. When the "Key Switch" is turned "ON", what does this allow you to do?

____

____

____

Your Notes:

Your Notes:

Section Nine

Your Notes:

Training Daily Guide

Session Nine

During this segment, participants will:

1. Understand the work address system.
2. Understand prepatory and miscellaneous functions used in CNC programming.
3. Understand how G Codes groups work in program format.
4. Explain most common G and M codes in CNC language.
5. Locate common errors in CNC program.
6. Explain canned Z axis cycles.
7. Explain the function of G99/G98 initial/R planes.
8. Review questions on G and M codes.

CNC Code Language

Definitions:

Work Address Format
Addressing each work in a block by one or more characters which identify the meaning of the word.

Word
An ordered set of characters composed of a letter and some numeric value which may be used to cause a specific action of a CNC machine.

Variable Block Format
A format which allows the number of works in successive blocks to vary. A word address system is also a variable block format.

Block
A group of "words" considered as a unit and seperated form other such units by an End of Block (EOB) character.

Words and Numbers:

N-Word
A sequence number used to identify a block of information in your program.

Sequence Number
A number addressed by the letter "N" identifying the location of blocks in a program.

Examples:

N5.......;
N10......;
N15.....;

CNC Code Language

Prepatory Codes (G Function)
A number following address "G" determines the meaning of the command for the concerned block.

G codes are divided into the following types:

One Shot G Code
G Code which is only effective in the block which it is specified.

Modal G Code
G Code which is effective until another G code of the same group is specified.

Example:
G01 and G00 ar modal G codes in Group 01.

G01 X________
2________
G00 X________
2________

G Codes

GROUP		DESCRIPTION
G00	01	RAPID POSITIONING
G01	01	LINEAR INTERPOLATION - USES FEEDRATE
G02	01	CW CIRCULAR INTERPOLATION
G03	01	CCW CIRCULAR INTERPOLATION
G04	*	DWELL - USES P FOR TIME, 1 SEC = P1000
G06	*	POSITIONING ERROR DETECT OFF ... YASNAC ONLY
G09	*	EXACT STOP CHECK -- FOR SHARP EDGES
G12	*	CIRCLE CUTTING CW ID ONLY ... YASNAC ONLY
G13	*	CIRCLE CUTTING CCW ID ONLY ... YASNAC ONLY
G17	02	XY PLANE DESIGNATION
G18	02	XZ PLANE DESIGNATION
G19	02	YZ PLANE DESIGNATION
G20	06	INCH INPUT DESIGNATION
G21	06	METRIC INPUT DESIGNATION
G27	*	REFERENCE POINT RETURN CHECK
G28	*	RETURN TO REFERENCE POINT (MACHINE " 0 ")
G29	*	RETURN FROM REFERENCE POINT
G40	07	TOOL RADIUS COMPENSATION CANCEL
G41	07	TOOL RADIUS COMPENSATION LEFT
G42	07	TOOL RADIUS COMPENSATION RIGHT
G43	08	TOOL LENGTH COMPENSATION PLUS DIRECTION
G44	08	TOOL LENGTH COMPENSATION MINUS DIRECTION
G49	08	TOOL LENGTH COMPENSATION CANCEL
G45	*	TOOL OFFSET INCREASE
G46	*	TOOL OFFSET DECREASE

* - ONE SHOT CODE
1. ONLY G-CODES FROM THE SAME GROUP CAN CANCEL EACH OTHER
2. ONLY 1 G-CODE FROM THRE SAME GROUP IN A LINE

G Codes

GROUP		DESCRIPTION
G81	09	CANNED CYCLE # 1 FEED IN RAPID OUT SPOT DRILLING
G82	09	CANNED CYCLE # 2 FEED IN, DWELL, RAPID OUT CHAMFERING
G83	09	CANNED CYCLE # 3 DEEPHOLE DRILLING
G84	09	CANNED CYCLE # 4 TAPPING (PITCH * RPM * .98 = FEEDRATE)
G85	09	CANNED CYCLE # 5 FEED IN, FEED OUT BORING
G86	09	CANNED CYCLE # 6 FEED IN, SPINDLE STOP, RAPID OUT - BORING
G87	09	CANNED CYCLE # 7 FEED IN, SPINDLE STOP, MANUAL OUT
G88	09	CANNED CYCLE # 8 FEED IN, DWELL, SPINDLE STOP, MANUAL OUT
G89	09	CANNED CYCLE # 9 FEED IN, DWELL, FEED OUT, - BORING
G73	09	CANNED CYCLE # 10 HIGH SPEED PECK CYCLE
G74	09	CANNED CYCLE # 11 COUNTER TAPPING (PITCH * RPM * .98 = FEEDRATE)
G76	09	CANNED CYCLE # 12 FINE BORING CYCLE
G80	09	CANNED CYCLE CANCEL
G90	03	ABSOLUTE PROGRAMMING (USES ABSOLUTE ZERO)
G91	03	INCREMENTAL PROGRAMMING (POINT TO POINT)
G92	*	ABSOLUTE ZERO POINT PRESET (PART ZERO)
G94	05	FEEDRATE PER MINUTE
G95	05	FEEDRATE PER REVOLUTION
G98	10	RETURN TO INITIAL LEVEL IN CANNED CYCLE
G99	10	RETURN TO R-LEVEL IN CANNED CYCLE

* - ONE SHOT CODE

1. ONLY G-CODES FROM THE SAME GROUP CAN CANCEL EACH OTHER
2. ONLY 1 G-CODE FROM THRE SAME GROUP IN A LINE

M Codes

GROUP	DESCRIPTION	USES
M00	PROGRAM STOP, STOPS SPINDLE, COOLANT	SHIFT OR FLIP PART
M01	OPTIONAL STOP, STOPS SPINDLE AND COOLANT	BEFORE TOOL CHANGES
M02	END OF PROGRAM	LOOPTAPE ONLY
M03	SPINDLE CLOCKWISE	TURNS ON SPINDLE
M04	SPINDLE CCW	TURNS ON SPINDLE
M05	SPINDLE STOP	SEE M19 NOTE
M06	TOOL CHANGE	ONLY AT ATC POSITION
M07	OIL MIST	OPTIONAL FUNCTION
M08	COOLANT ON	FLOOD COOLANT
M09	COOLANT OFF	ALSO TURNS OFF M07
M18	NEUTRAL	FOR GEAR DRIVEN HEADS
M19	SPINDLE ORIENT	WILL SHUT OFF SPINDLE USED W/ZERO RET Z AXIS
M20	SPINDLE ORIENT OFF	USUALLY FOR OPERATOR USE ONLY
M21-28	OPTIONAL M-FUNCTIONS	STANDARD
M29	TOOL POCKET VERTICAL POSITION	DOUBLE ARM MACHINES ONLY
M30	END OF PROGRAM	USED AT END OF MEMORY
M42	LOW RANGE SPEED RUN IN HIGH GEAR	TAPPING, YASNAC ONLY
M46	CLEARS TOOL POCKET AT END OF PROGRAM	

151

M Codes

GROUP	DESCRIPTION	USES
M48	FEEDRATE OVERRRIDE EFFECTIVE	
M49	FEEDRATE OVERRIDE CANCEL LOCK FEEDRATE AT 100 %	
M73	X - AXIS MIRROR IMAGE ON	FANUC ONLY
M74	Y - AXIS MIRROR IMAGE ON	FANUC ONLY
M76	4TH AXIS MIROR IMAGE ON	FANUC ONLY
M77	MIRROR IMAGE CANCEL	FANUC ONLY
M94	MIRROR IMAGE OFF	YASNAC ONLY
M95	MIRROR IMAGE ON	YASNAC ONLY
M98	JUMP COMMAND INTO SUB-PROGRAM	MULTIPLE OPERATIONS
M99	JUMP BACK INTO MAIN PROGRAM	

Error Exercise

```
O56432; ( NINE ERROR PROGRAM )
N1 G0 G20 G90 G40 G80
N5 T1 M6 ( .5 DIA RH CUT DRILL )
N10 S1000 M4 M8
N15 G54 X1. Y-1.
N20 G43 Z.1 H0
N25 G82 X1.0 R.1 Z-.3 P100 F50
N30 X3.
N35 X6.
N40 X9
N45 G0 G28 G91 Z0 M
N50 G28 X0 Y0 M0
N55 M10
```

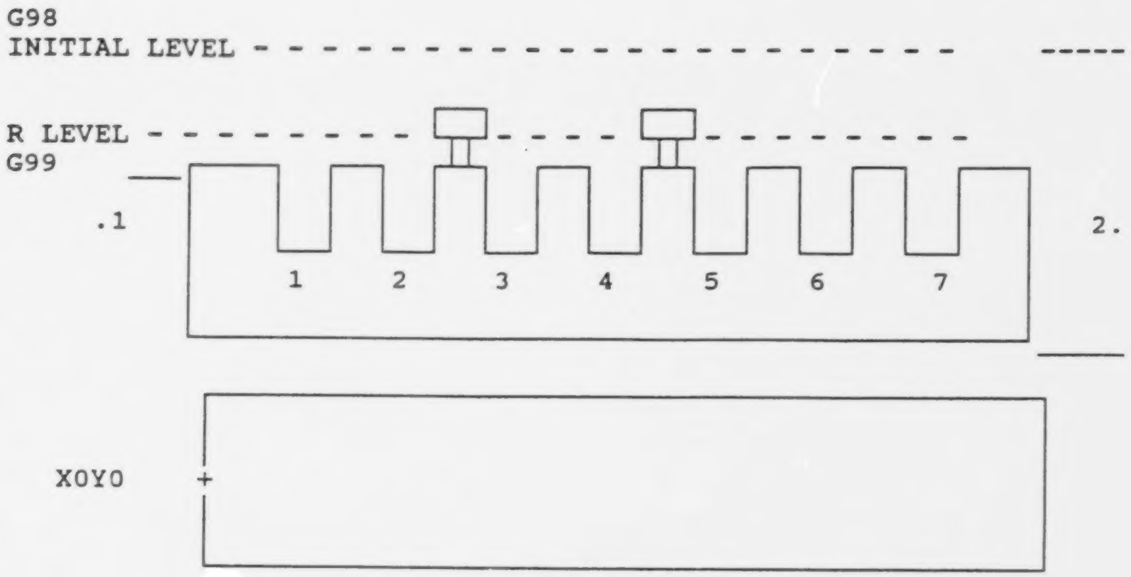
LIST THE ERRORS FOUND BELOW AND CORRECTION.

ERROR	CORRECTION
1. ____________________	____________________
2. ____________________	____________________
3. ____________________	____________________
4. ____________________	____________________
5. ____________________	____________________
6. ____________________	____________________
7. ____________________	____________________
8. ____________________	____________________
9. ____________________	____________________

Canned Z Axis Cycles

G #	PLUNGING	AT HOLE BOT	RETRACTION	APPLICATION
G73	WOODPECKER FEED	___________	RAPID TRAVERSE	HIGH SPEED DEEP HOLE DRILLING
G74	FEED	SPINDLE FWD AFTER DWELL	SPINDLE REVERSE AFTER FEED	REVERSE TAPPING
G76	FEED	SPINDLE INDEX- SHIFT	RAPID TRAVERSE SHIFT SP. START	BORING
G77	SPINDLE INDEX RAPID	DWELL	RAPID TRAV. SPINDLE INDEX	BACK BORING
G80	--------	----------	-----------	CANCEL
G81	FEED	----------	RAPID TRAVERSE	DRILLING
G82	FEED	DWELL	RAPID TRAVERSE	SPOT, FACING
G83	WOODPECKER	----------	RAPID TRAVERSE	DEEP HOLE DRILLING
G84	FEED	SPINDLE REV AFTER DWELL	SPINDLE FWD RUN AFTER FEED	TAPPING
G85	FEED	----------	FEED	BORING
G86	FEED	SPINDLE STOP	RAPID TRAVERSE SPINDLE START	BORING
G87	FEED	SPINDLE STOP	MANUAL RETRACT SPINDLE START	BORING
G88	FEED	SPINDLE STOP AFTER DWELL	MANUAL RETRACT SPINDLE START	BORING
G89	FEED	DWELL	FEED	BORING

G99 R-Level/G98 Initial Level


G98
INITIAL LEVEL
R LEVEL
G99
.1
2.
1
2
3
4
5
6
7
X0Y0

NOTE: DRILL MUST CLEAR PROJECTIONS BETWEEN HOLES TWO AND THREE AND FOUR AND FIVE.

```
O2255
N1 G0G90G40G80G20
N5 T1M6
N10 S3000 M3
N15 G54 X1.0Y0
N20 G43 Z 2. H01 ______________________________

N25 G99 G81 Z-1.R.1 F8. _________________________

N30 G98 X2.
N35 G99 X3.
N40 G98 X4.
N45 G99 X5.
N50 X6.
N55 X7.
N45 G80 M9
N50 G91 G28 Z0 G0
N55 G28 X0 Y0
N60 M30
```

G & M Code Questions

1. Which three G Codes have to do with tool radius compensation?
   A] ____________________
   B] ____________________
   C] ____________________

2. Which G Code cancels any canned Z axis cycle?
   A] ____________________

3. Which reference plane is set in the program by a rapid (Gøø) command followed by a Z axis position?
   A] ____________________

4. Which three G Codes have to do with tool length compensation?
   A] ____________________
   B] ____________________
   C] ____________________

G & Mode Code Questions

5. Which G code is the standard start-up condition for feed rates on CNC milling machines?

   A] ____________________

6. Which G codes could cancel a G01 command?

   A] ____________________
   B] ____________________
   C] ____________________

7. What is a one-shot G code?

   A] ____________________

8. How many G codes are in Group #9?

   A] ____________________

G & Mode Code Questions

9. How many different group of G codes are there?

A] ____________________

10. A M09 cancels which M codes?

A] ____________________
B] ____________________

11. Explain what happens when a M01 is commanded.

A] ____________________
____
____

12. Which M code would be used for left-hand cut cutting tools?

A] ____________________

Your Notes:

Your Notes:

Section Ten

Your Notes:

Training Daily Guide

Session Ten

During this two hour segment, participants will:

1. Understand components of the Niigata CNC horizontal machine center
2. Understand main and sub operating panels
3. Know how to input tool length offsets
4. Know how to zero return the axes to machine home
5. Understand MDi operations
6. Understand program entry and editing
7. Know how to use soft keys for program manipulation
8. Understand operation of APC and ATC units on machines
9. Know how to recover from a feed hold(example: tool breakage)
10. Understand procedure to mount and dismount tools in spindle and tool magazine

more on next page

Training Daily Guide

11. Understand use of screen displays

12. Know where to look for typical alarm codes and how to explain the reasons for alarms

13. Recognize a sub-program which is used to make the machine go to a safe home position prior to starting the main program.

Machine Specs

Niigata CNC Horizontal M.C.

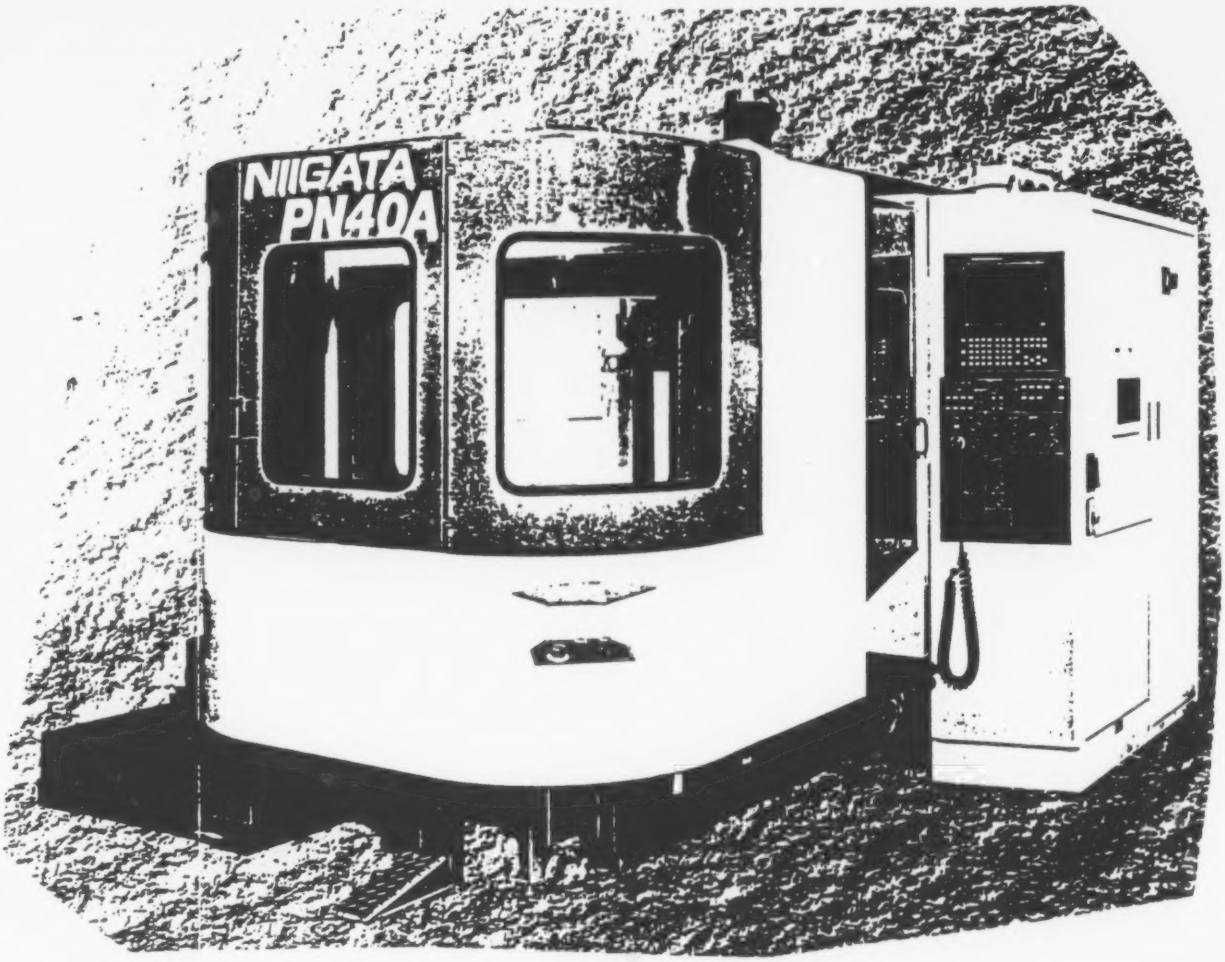
NIIGATA

MACHINE SPECIFICATIONS

	Item	Metric	Inch
TRAVEL & WORK CAPACITY	X axis column travel	610 mm (330 + 280)	24.0" (13.0" + 11.0")
	Y axis vertical head travel	500 mm	19.7"
	Z axis table travel	560 mm	22.0"
	Spindle center line to pallet surface	50–550 mm [150–650 mm]	1.97–21.6" [5.9"–25.6]
	Spindle nose to table center line.	150–710 mm	5.9–27.9"
TABLE	Table working surface	400 × 400 mm	15.7" × 15.7"
	Table increments	1° [0.001°]	1° [0.001°]
	Maximum load on pallet	400kg	880 lbs.
SPINDLE	Spindle drive motor	AC11/7.5kw	AC15/10 HP
	Spindle speeds	12,000min.$^{-1}$ (rpm)	12,000 rpm
	Spindle max. torque	292 N·m	215 ft·lbs.
	Spindle taper	No. 40	No. 40
FEEDRATE	Rapid traverse X·Y·Z	24 m/min.	945 ipm
	Cutting X·Y·Z	1–15,000 mm/min.	0.04–590 ipm
	Table index speed/90°	2.5 sec.	2.5 sec.
AUTOMATIC TOOL CHANGER (ATC)	Tool magazine capacity	30 [40/60/90/120/185]	30 [40/60/90/120/185]
	Tool selection	Short cut random	Short cut random
	Tool shank	BT40	CT40
	Maximum tool length	350 mm	13.8"
	Maximum milling cutter dia.	95 mm	3.7"
	*(adjacent pockets empty)	200 mm	7.9"
	Maximum tool mass	10kg	22 lbs.
	Tool change time (tool to tool)	1.4 sec.	1.4 sec.
AUTOMATIC PALLET CHANGER (APC)	Type	Direct rotary	Direct rotary
	Pallet change time	5.5 sec.	5.5 sec.
	Number of pallet	2 [6/8/10/12]	2 [6/8/10/12]
ACCURACY	Positioning/full stroke X·Y·Z	±0.003 mm	±0.00012"
	Positioning with scales	±0.002 mm	±0.00008"
	Repeatability X·Y·Z	±0.0015 mm	±0.00006"
	Repeatability with scales	±0.001 mm	±0.00004"
	Table index	±2.5 sec.	±2.5 sec.
GENERAL	Base machine mass approx.	9,000kg	20,000 lbs.
	Machine space W/D/H approx.	2,460/3,700/2,467 mm	97"/146"/97"
	Floor to table surface	1,016 mm	40"
	Power	45 KVA	45 KVA

- [ ] Optional specifications
- Product specifications, accessories and machine appearance are subject to change without notice

Niigata CNC Horizontal M.C.


NIIGATA
PN40A

BEST COPY AVAILABLE

Niigata CNC Horizontal M.C.
Fanuc Control Specs

CONTROL SPECIFICATIONS (FANUC-[illegible])

STANDARD

- Simultaneous controllable axes: Three axes (X,Y,Z)
- Least input increment: 0.001mm (0.0001")
- Least command increment: 0.001mm
- Linear interpolation GO1
- Multi-quadrant circular interpolation
- Feedrate command: Direct designation of mm/min or inch/min feed.
- Feedrate override: 0 ~ 200% (10% increments)
- Tangential speed constant control
- Cutting feedrate clamp
- Override cancel
- Automatic acceleration/deceleration
- Exact stop/cutting mode G09, G61, G63, G64
- Dwell
- Reference point return: Manual, automatic G27, G28, G29)
- Machine coordinate system selection: G53
- Work coordinate system selection: 6 types (G54~G59)
- Local coordinate system setting: G52
- Work coordinate system change: G92
- Absolute/incremental programing
- Decimal point input
- Program number search
- Program number: 4 digits/Program name : 16 characters
- Sequence number search
- Main program/subprogram Subprogram/4 folds nested
- Tape code: E1A, RS244, ISO-840 automatic recognition
- Label skip
- Control in/out
- Optional block skip
- ISO code input
- Tool offset selection by T code 99
- Radius designation on arc
- 2nd auxiliary function
- Tool length compensation G43, G44
- Tool offset G45~G48
- Cutter compensation C G40~G42
- Tool offset amount memory A:
  ±6 digits, common to all tools, 32 pairs
- Backlash compensation:
  Editing during automatic operation
- Registerable programs:
  100 (Program name display is possible)
- Part program storage length 80m (262ft)
- Reader/Puncher interface:
  RS232 × 1, RS422 × 1, RS232C or 20mA current loop" × 1
- Keyboard type MDI/CRT character display
  9 monochrome CRT
- Data protection key:
  3 types
- Self-diagnosis function
- Follow-up:
  At emergency stop
- Servo off
- Buffer register
- Manual continuous feed
- Incremental feed
- Manual handle feed
- Manual absolute on/off
- All axis Machine lock
- Machine lock on each axis
- Auxiliary function lock
- Dry run
- Single block
- Over travel
- Stored stroke check 1
- Interlock
- Stored pitch error compensation
- Clock function

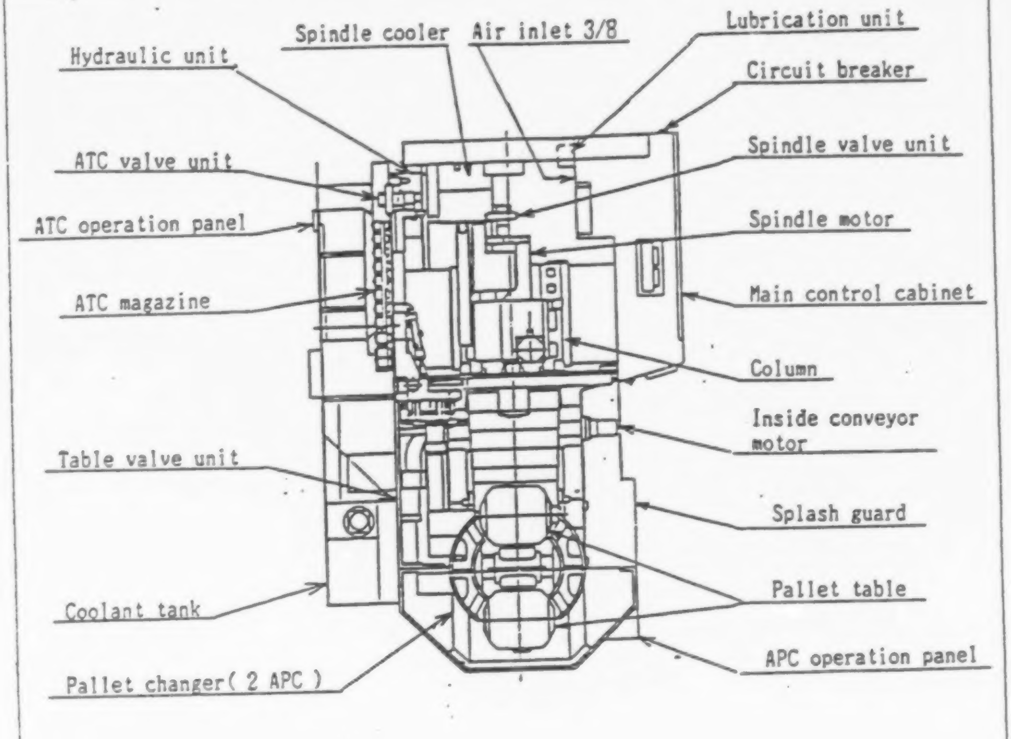
OPTIONAL

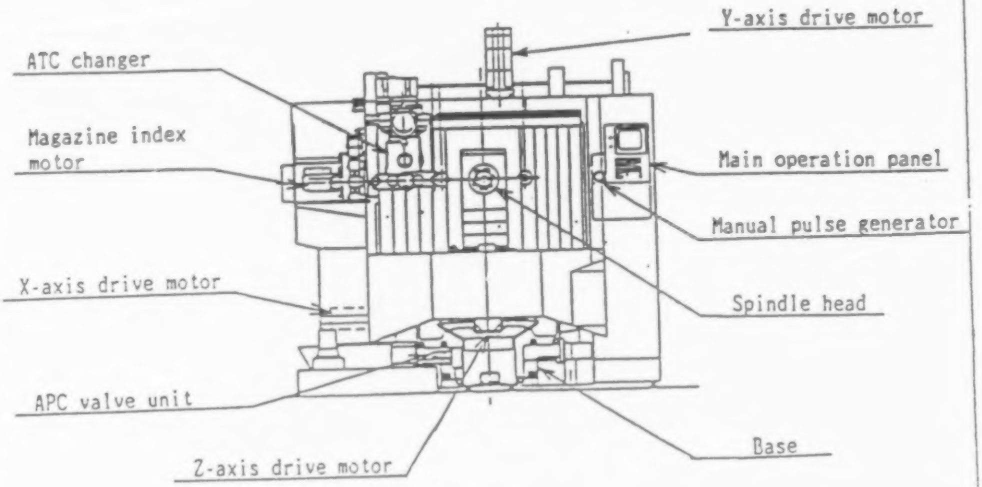
- 14" color CRT
- Part program storage length
  320, 640, 1280, 2560, 5120m
- Tool offset selection by T code 200, 499, 99
- Tool offset amount memory C
- Registered program
  200, 400, 1000
- External data input/output
- Helical interpolation
- Single direction positioning
- Conversational automatic programming function incl. 14 inch CRT and Graphic display
- Automatic corner override
- Handy file
- Sequence No. comparison and stop
- Tool life management
- Program restart
- High speed skip signal input
- Stored stroke check 2
- Tool length measurement
- Programmable data input G10L1
- Scaling
- Linear acc/dec before cutting feed interpolation
- Polar coordinate command
- Optional angle chamfering corner R
- Programmable mirror image
- Interruption type custom macro
- Coordinate system rotation
- Additional axes:
  Max. 5 axes
- Simultaneously controllable axes expansion
  Max. 5 axes
- RUN hour display
- Background editing:
  Editing during automatic operation

BEST COPY AVAILABLE

MACHINE COMPOSITION & UNIT NAME

§1. OUTLINE	MACHINE COMPOSITION & UNIT NAME		PN40


Spindle cooler
Air inlet 3/8
Lubrication unit
Hydraulic unit
Circuit breaker
ATC valve unit
Spindle valve unit
ATC operation panel
Spindle motor
ATC magazine
Main control cabinet
Column
Inside conveyor motor
Table valve unit
Splash guard
Pallet table
Coolant tank
APC operation panel
Pallet changer(2 APC)

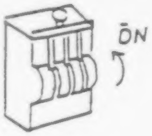

Y-axis drive motor
ATC changer
Magazine index motor
Main operation panel
Manual pulse generator
X-axis drive motor
Spindle head
APC valve unit
Base
Z-axis drive motor

	NIIGATA ENGINEERING CO., LTD.		T-100

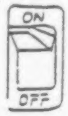
TURN ON PROCEDURE OF CONTROL

2. Turning ON Procedure of Control Power

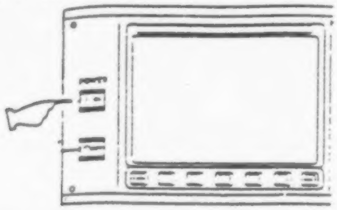
①. Turn ON primary power to the machine.



②. Turn ON Circuit Breaker.



③. Turn ON Control Power of NC Unit.



Few seconds later after power ON, Position Screen is displayed.

④. Turn ON Control Power of Machine Operation Panel.

Hydraulic pump is started, and Power Indicating Lamp is lit.

Note-1 : If Hydraulic pump is not started despite Control Power button is depressed, check whether EMG Button of either ATC, APC Operation Panel is kept ON. If kept ON , click EMG button CW direction light for reset.

Note-2 : Hydraulic pump is actuated during Control Power button is depressed, and deactuated and stopped immediately after depressing :

Either of X, Y and Z-axis has possibility of Overrun (2nd Overrun).

* Recovery procedure :

Keep depressing Control Power Button ON, and return overrun axis its inside of stroke range by MPG Handle.

Then depress NC RESET button.

BEST COPY AVAILABLE

TURN OFF PROCEDURE OF CONTROL

3. Turning OFF Procedure of Control Power

①. Considering about machine balance and easier zero return operation, move each axis to its stroke center area as long as each axis has no interference with workpiece on the pallet.

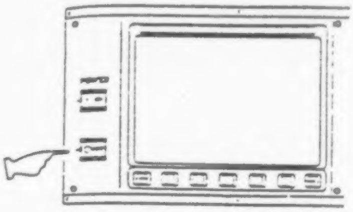
Mode selection switch " RAPID " is selected. → Axis moving by 「＋」 or 「－」 button (Refer to P.   Rapid Traverse in details.)

②. Depress Mushroom type EMG button.

Lamp for 「 CONTROL ON 」 indicating is turned OFF, and hydraulic pump is stopped.

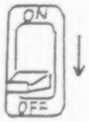
Turn EMG button CW direction little light for button reset.

③. Depress OFF button of NC-unit.

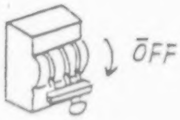


Screen display is turned OFF.

④. Turn OFF Circuit Breaker of Main Control Cabinet.



⑤. Turn OFF the power to the machine.

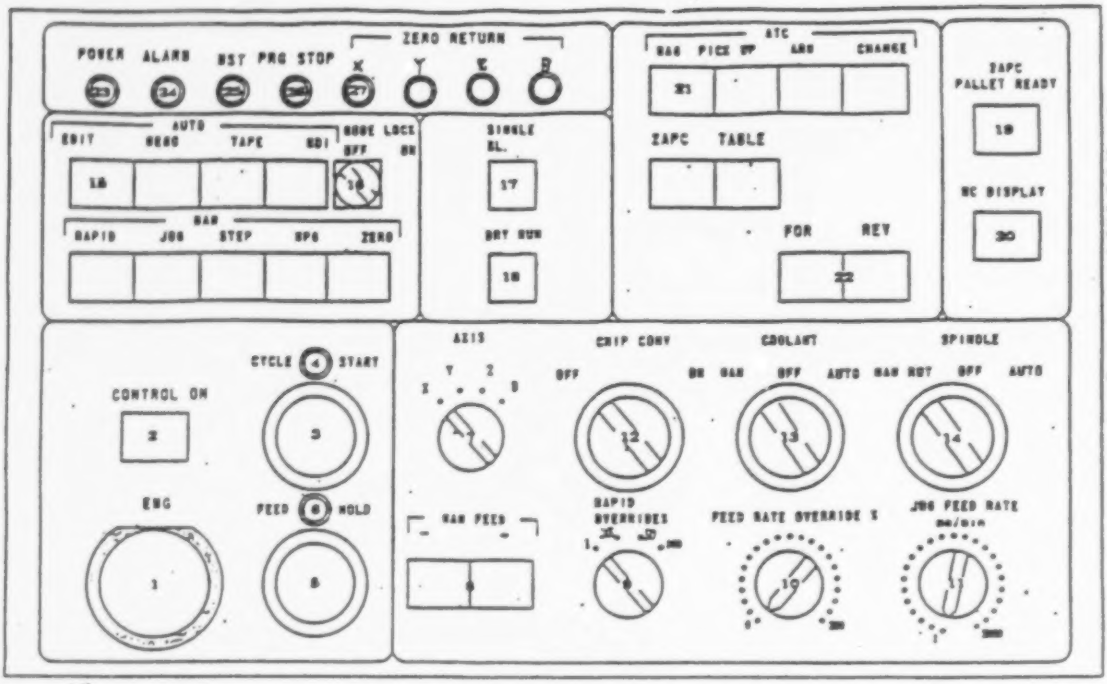


NIIGATA CNC HMC

MAIN OPERATING PANEL

PN 40 Operating Panel

Main Operating Panel


POWER
ALARM
RST
PRG STOP
ZERO RETURN
X
Y
Z
B
23
24
25
26
27
AUTO
EDIT
MEMO
TAPE
MDI
MODE LOCK
OFF
ON
15
16
MAN
RAPID
JOG
STEP
HPG
ZERO
SINGLE BL.
17
DRY RUN
18
ATC
MAN
PICK UP
ARM
CHANGE
21
2APC
TABLE
FOR
REV
22
2APC PALLET READY
19
NC DISPLAY
20
CONTROL ON
2
CYCLE
4
START
3
EMG
1
FEED
6
HOLD
5
AXIS
X
Y
Z
B
7
CHIP CONV
OFF
ON
12
COOLANT
MAN
OFF
AUTO
13
SPINDLE
MAN RET
OFF
AUTO
14
MAN FEED
-
+
8
RAPID OVERRIDES
1
25
50
100
9
FEED RATE OVERRIDE %
0
200
10
JOG FEED RATE
mm/min
1
2000
11

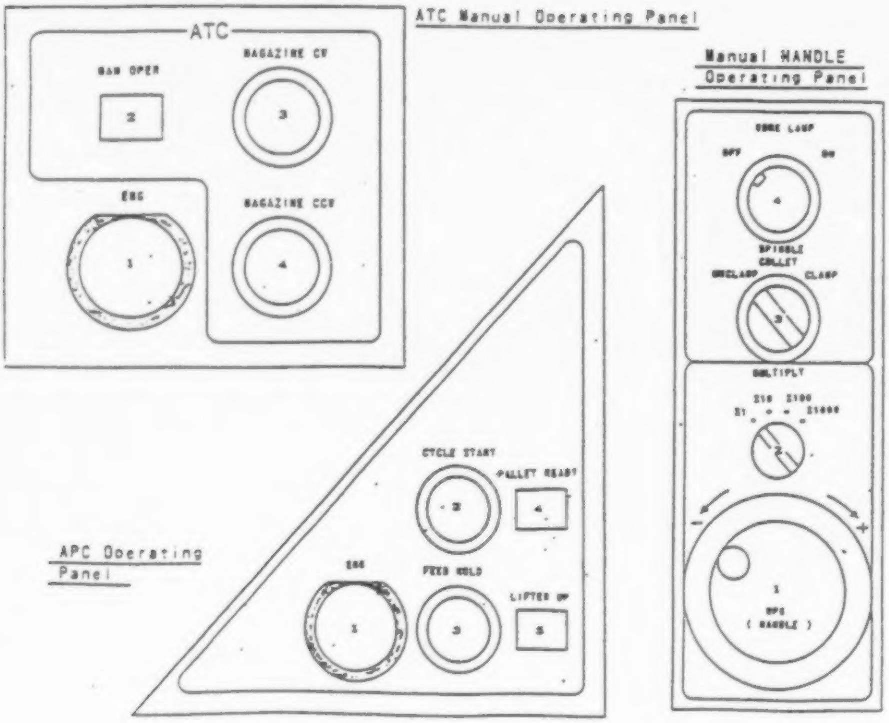
NIIGATA CNC HMC

MAIN OPERATING PANEL

NAME THE CONTROL FEATURES

1. ____________________
2. ____________________
3. ____________________
4. ____________________
5. ____________________
6. ____________________
7. ____________________
8. ____________________
9. ____________________
10. ____________________
11. ____________________
12. ____________________
13. ____________________
14. ____________________
15. ____________________
16. ____________________
17. ____________________
18. ____________________
19. ____________________
20. ____________________
21. ____________________
22. ____________________
23. ____________________
24. ____________________
25. ____________________
26. ____________________
27. ____________________

ATC MANUAL OPERATING PANEL
APC OPERATING PANEL
MANUAL HANDLE OPERATING PANEL


ATC Manual Operating Panel
ATC
MAN OPER
2
MAGAZINE CW
3
EMG
1
MAGAZINE CCW
4
Manual HANDLE Operating Panel
WORK LAMP
OFF
ON
4
SPINDLE COLLET
UNCLAMP
CLAMP
3
MULTIPLY
X10 X100
X1 X1000
2
−
+
1
MPG (HANDLE)
APC Operating Panel
CYCLE START
2
PALLET READY
4
EMG
1
FEED HOLD
3
LIFTER UP
5

NIIGATA CNC HMC

EXPLAIN THE FUNCTION OF EACH FEATURE

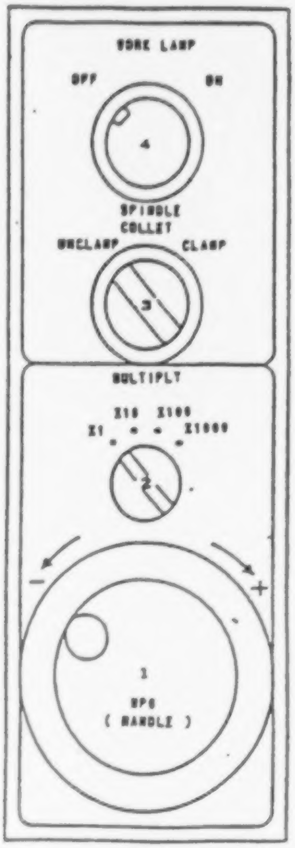
MANUAL HANDLE

MPG (1) ______________________________

MULTIPLY (2) ______________________________

SPINDLE COLLET (3) ______________________________

WORKLAMP (4) ______________________________


WORK LAMP
OFF
ON
4
SPINDLE COLLET
UNCLAMP
CLAMP
3
MULTIPLY
X10
X100
X1
X1000
2
-
+
1
MPG (HANDLE)

NIIGATA CNC HMC

EXPLAIN THE FUNCTION OF EACH FEATURE

AUTOMATIC PALLET CHANGER

EMG (1) ______________________________

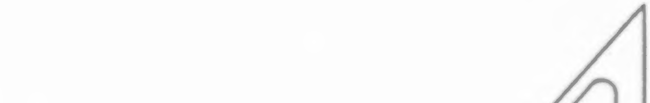
CYCLE START (2) ______________________________

FEED HOLD (3) ______________________________

PALLET READY (4) ______________________________

____

LIFTER UP (5) ______________________________


CYCLE START
PALLET READY
2
4
EMG
FEED HOLD
LIFTER UP
1
3
5

NIIGATA CNC HMC

EXPLAIN THE FUNCTION OF EACH FEATURE

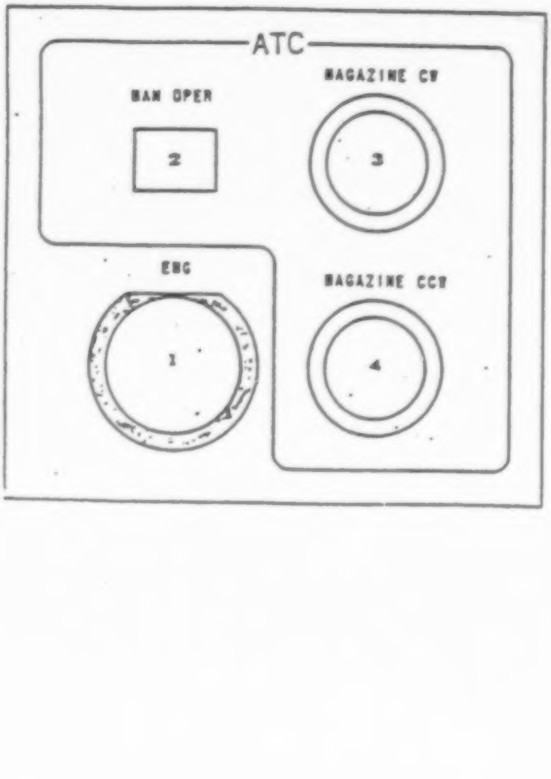
AUTOMATIC TOOL CHANGER

EMG (1) ______________________________

MAN OPER (2) ______________________________

MAGAZINE CW (3) ______________________________

MAGAZINE CCW (4) ______________________________


ATC
MAN OPER
2
MAGAZINE CW
3
EMG
1
MAGAZINE CCW
4

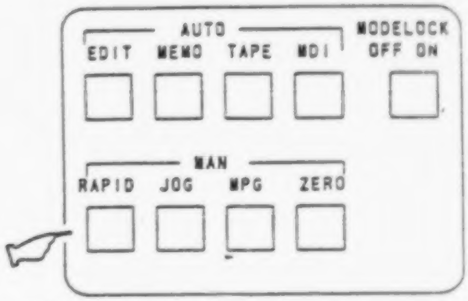
MOVING THE AXES USING RAPID

5. Rapid Traverse, JOG Feed & Manual Pulse Generator (MPG)

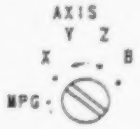
5-1. Rapid Traverse

Each axis is moved to any desired position by rapid traverse.

①. Depress 「RAPID」 button of mode section switch.

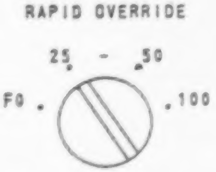

AUTO
EDIT
MEMO
TAPE
MDI
MODELOCK
OFF ON
MAN
RAPID
JOG
MPG
ZERO

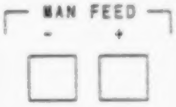
②. Select desired axis to be moved on AXIS selecter.


AXIS
Y Z
X
B
MPG

③. Depress 「+」 or 「−」 button of MAN FEED.

Axis moves during button depressing.
Moving speed is set by Rapid Override.


RAPID OVERRIDE
25 - 50
F0
100



Note :

If an axis moves out of stroke limit(overrun) accidentally, move the axis to be in the stroke range.

If 「 CONTROL ON 」 indicating lamp is turned OFF due to overrun, recover with the following procedures.

ⅰ). Depress 「 CONTROL ON 」 button and keep depressing it.

ⅱ). Move overrun axis to the opposite direction by MPG handle.
(Refer to Item 5-3. Manual Pulse Generator / P.8)

ⅲ). Keep OFF from 「 CONTROL ON 」 button.

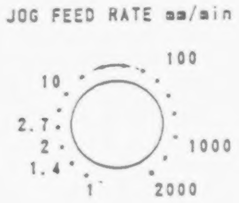
ⅳ). Depress RESET button of NC unit. (OVERRUN indication signal is turned OFF on the screen.)

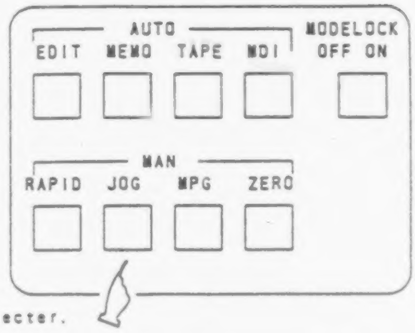
BEST COPY AVAILABLE

MOVING THE AXES USING JOG OR MPG

5-2. JOG Feed

Each axis can be moved by set feed for JOG Feed manually.


JOG FEED RATE mm/min
100
10
2.7
2
1.4
1
1000
2000


AUTO
MODELOCK
EDIT
MEMO
TAPE
MDI
OFF ON
MAN
RAPID
JOG
MPG
ZERO

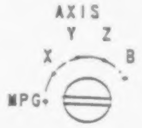
①. Depress 「 JOG 」 button of mode selecter.

②. Set jog feed rate to desired speed.

③. Select desired axis to be moved on AXIS selecter.
Depress 「+」 or 「−」 button of MAN FEED as in case of RAPID Traverse.

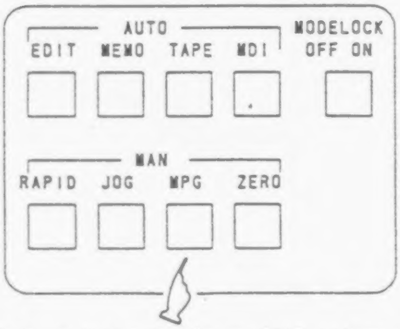
5-3. Manual Pulse Generator (MPG) / Handle feed

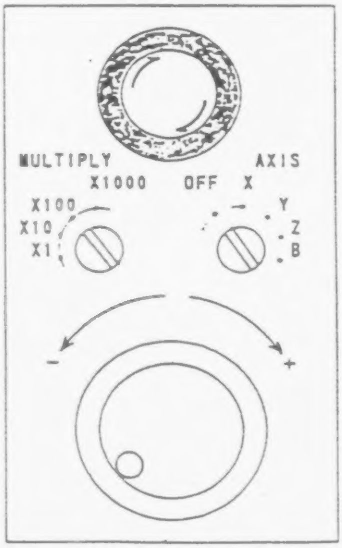
Each axis can be moved to desired position by Handle (Manual Pulse Generator : MPG).


AXIS
Y
Z
X
B
MPG

①. Select 「 MPG 」 position of axis selecter.

②. Depress 「 MPG 」 button of mode selecter.


AUTO
MODELOCK
EDIT
MEMO
TAPE
MDI
OFF ON
MAN
RAPID
JOG
MPG
ZERO


MULTIPLY
X1000
X100
X10
X1
AXIS
OFF
X
Y
Z
B
−
+

③. Select desired axis on AXIS selecter of MPG operation box right.

④. Set Desired position of MULTIPLY switch of MPG operation box right to either X1, X10, X100 or X1000.

X1 = 0.001 mm per 1DIV.
X10 = 0.01 mm per 1DIV.
X100 = 0.1 mm per 1DIV.
X1000 = Same as X100

BEST COPY AVAILABLE

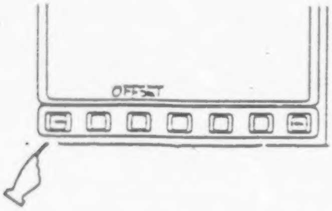
INPUT OF TOOL OFFSETS

8. Tool Offset

Tool Length and Tool Dia. can be input to display on CRT screen for Tool Length Offset and Tool Dia. Offset compensation.

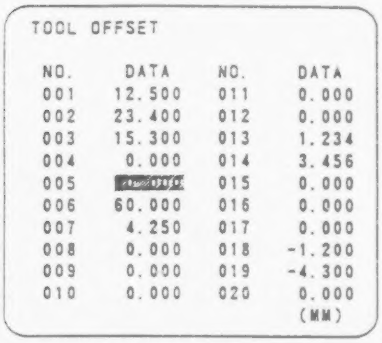
①. Display OFFSET screen to CRT as follows.

i).



Depress function MENU key located at bottom left to display ' OFFSET'.

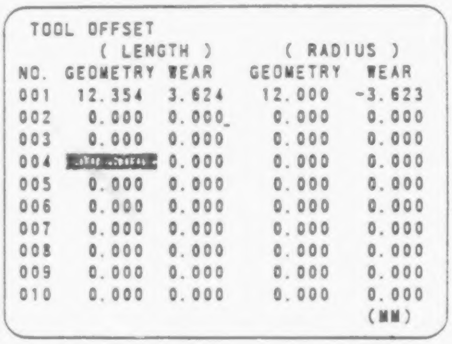
ii). Depress OFFSET soft key. Then the following screen is displayed. If not displayed, depress the same OFFSET key few more times.



TOOL OFFSET

NO.	DATA	NO.	DATA
001	12.500	011	0.000
002	23.400	012	0.000
003	15.300	013	1.234
004	0.000	014	3.456
005	[illegible]	015	0.000
006	60.000	016	0.000
007	4.250	017	0.000
008	0.000	018	-1.200
009	0.000	019	-4.300
010	0.000	020	0.000
			(MM)

Standard screen



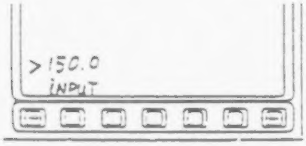
TOOL OFFSET

	(LENGTH)		(RADIUS)	
NO.	GEOMETRY	WEAR	GEOMETRY	WEAR
001	12.354	3.624	12.000	-3.623
002	0.000	0.000_	0.000	0.000
003	0.000	0.000	0.000	0.000
004	[illegible]	0.000	0.000	0.000
005	0.000	0.000	0.000	0.000
006	0.000	0.000	0.000	0.000
007	0.000	0.000	0.000	0.000
008	0.000	0.000	0.000	0.000
009	0.000	0.000	0.000	0.000
010	0.000	0.000	0.000	0.000
				(MM)

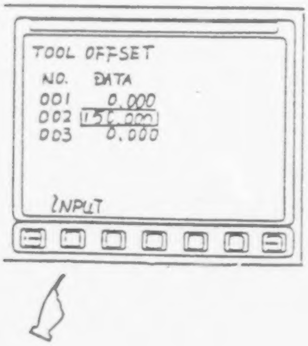
Screen with Offset Memory 'C'(OP.)

②. Move Cursor to desired position using PAGE keys(↑ ↓) and CURSOR move keys (← [illegible] . ← →).

③. Key in desired numeric value as shown below.



④. Depress ' INPUT ' soft key. Offset value is input or altered.



TOOL OFFSET

NO. DATA

001 0.000

002 150.000

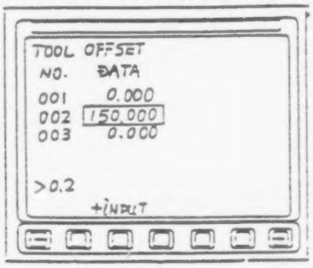
003 0.000

ADDING TO OFFSET VALUE

REFERENCE

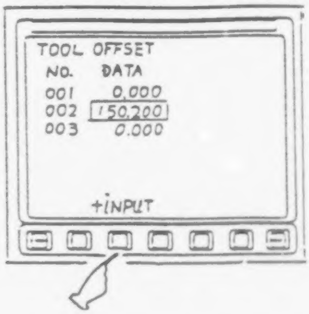
1). When Offset value is addtionally added to the present offset value :

①. Move cursor to desired position and key in additional value.



```
TOOL OFFSET
 NO.   DATA
 001    0.000
 002  150.000
 003    0.000

>0.2
      +INPUT
```

②. Depress ' + INPUT ' soft key, and above additional value is added.

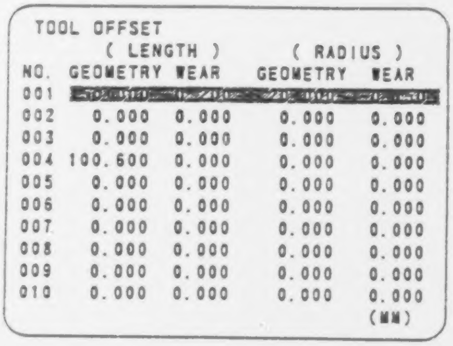


```
TOOL OFFSET
 NO.   DATA
 001    0.000
 002  150.200
 003    0.000

      +INPUT
```

When +INPUT as -0.2, offset value is reduced to 149.800 from 150.000

TOOL OFFSET MEMORY " C "

2). Tool Offset Memory 'C' (Option) :



TOOL OFFSET

NO.	(LENGTH) GEOMETRY	(LENGTH) WEAR	(RADIUS) GEOMETRY	(RADIUS) WEAR
001	[illegible]	[illegible]	[illegible]	[illegible]
002	0.000	0.000	0.000	0.000
003	0.000	0.000	0.000	0.000
004	100.600	0.000	0.000	0.000
005	0.000	0.000	0.000	0.000
006	0.000	0.000	0.000	0.000
007	0.000	0.000	0.000	0.000
008	0.000	0.000	0.000	0.000
009	0.000	0.000	0.000	0.000
010	0.000	0.000	0.000	0.000

(MM)

Both of Tool Length Offset (H-code) and Tool Radius Offset (D-code) using the same Offset No. can be used. and also GEOMETRY and WEAR Offset can be set together as shown left.

GEOMETRY : Tool geometry offset

WEAR : Tool wear offset

H1 works as offset value : 150.0 + 0.2 = 150.2

D1 works as offset value : 20.0 + (-0.15) = 19.85

ZERO RETURN METHOD

4. Zero Return Method (Reference Point Return)

After turning ON the power, Zero Return (Reference Point Return) must be required.

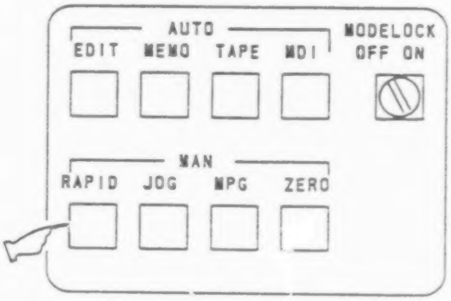
Zero Point (,Reference Point) of each axis is PLUS (+) stroke end.

4-1. Zero Return of X, Y, Z and B-axis (MC-table : OP.)

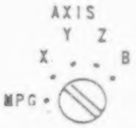
● When an axis is on Zero position or near by Zero position :

①. Depress 「RAPID 」 button on Mode selector switches.

Green lamp is lit.


AUTO
EDIT
MEMO
TAPE
MDI
MODELOCK
OFF ON
MAN
RAPID
JOG
MPG
ZERO

②. Select desired axis (X, Y, Z or B) to move on axis selection switch.

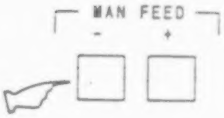

AXIS
Y
Z
X
B
MPG

③. Depress 「 − 」 button of axis movement.

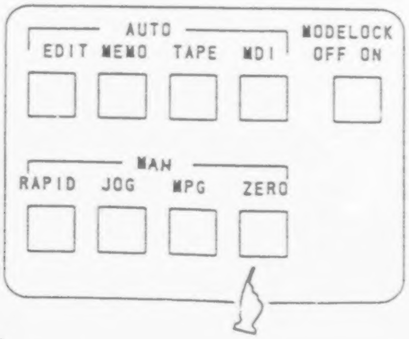
During keeping to depress, the axis keeps to move and stops by releasing.

Above ② & ③ operations are repeated about axes near by or on zero position.

Moving distance : Approximately 100 mm



④. Depress 「 ZERO 」 button on mode selection switch.


AUTO
EDIT
MEMO
TAPE
MDI
MODELOCK
OFF ON
MAN
RAPID
JOG
MPG
ZERO

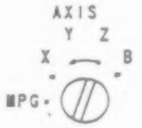
ZERO RETURN METHOD

● When an axis is 100 mm or more distance from its zero point :

Execute from above Item ④ for zero return on required axis.
Repeat the same procedure for another axis if any.

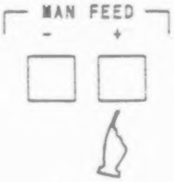
⑤. Select desired axis to be returned.

Z-AXIS MUST BE SELECTED 1ST to avoide tool and work or fixture interference. If either X, Y, or B-axis is selected, tool may interfer with work or fixture.



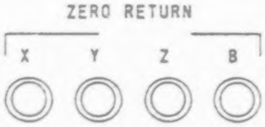
⑥. Depress 「＋」 button of axis movement.

Selected axis moves to zero point in plus (+) direction, and decelerated when it reaches close to the zero point.
Keep depressing the button until reaching this deceleration range.
Zero position indicating lamp is lit for the axis in this operation.



Repeat ⑤ & ⑥ operation for each axis.

Make sure zero return lamps are lit for X, Y, and Z-axis(B-axis : NC-table)

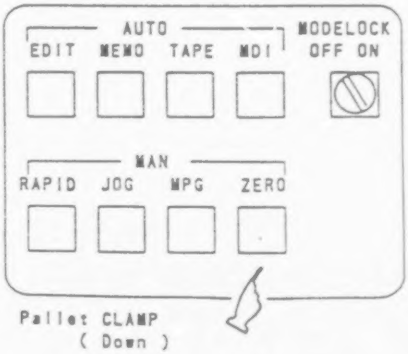


4-2. Zero return of B-axis (Either 72 or 360 position index table)

①. Depress 「 ZERO 」 button on mode selection switch.

②. Select「B」 on axis selecter.

③. Depress 「＋」 button of axis movement.



Motion :

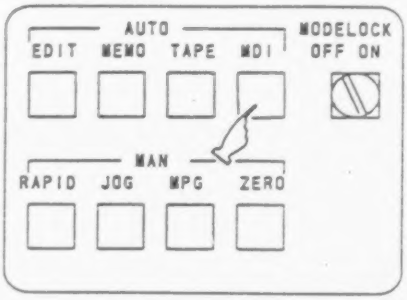
Pallet UNCLAMP → 0° indexing → Pallet CLAMP
(Lifted.) (Down)

MDI OPERATION

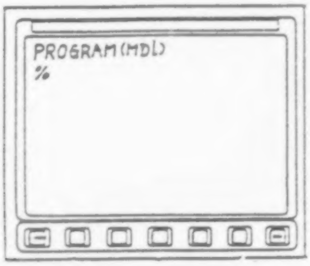
6. MDI (Manual Data Input) Operation

When either motion of Spindle rotation, Tool change, Pallet change or simple axis movement is desired, motion program is input manually to NC unit and executed automatically.

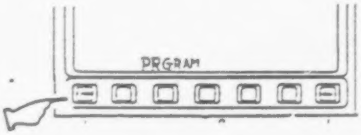
①. Depress 「 MDI 」 button as shown right.


AUTO
EDIT
MEMO
TAPE
MDI
MODELOCK
OFF ON
MAN
RAPID
JOG
MPG
ZERO

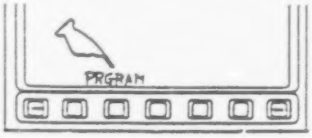
②. Check display of top left on the CRT screen.
To display PROGRAM (MDI) on the screen ;


PROGRAM(MDI)
%

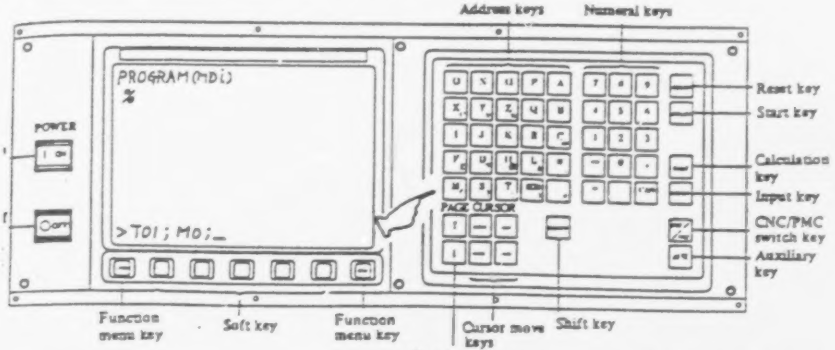
i). Depress left function MENU key, and display PRGRAM as shown.



ii). Depress PRGRAM soft key. PROGRAM (MDI) is shown at left top corner of the screen. If not displayed, repeat to depress it again until displaying.



③. Key in desired command using address keys and numeric keys.


Address keys
Numeral keys
POWER
PROGRAM(MDI)
%
>T01; M0;_
PAGE CURSOR
Reset key
Start key
Calculation key
Input key
CNC/PMC switch key
Auxiliary key
Function menu key
Soft key
Function menu key
Page keys
Cursor move keys
Shift key

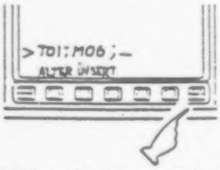
BEST COPY AVAILABLE

MDI OPERATION

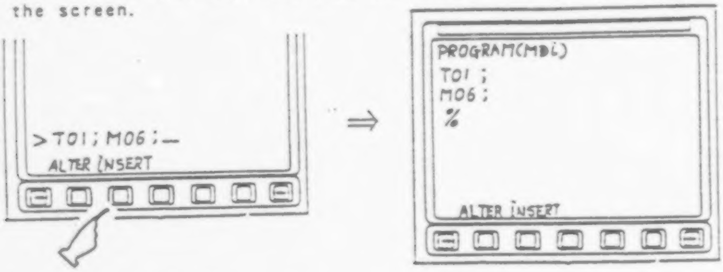
Keyed in command is displayed on the corner left below of the screen as shown.

Ex. > T 0 1 ; M 0 6 ; _

④. Depress right function MENU key, and keep depressing until displaying 'INSERT'.

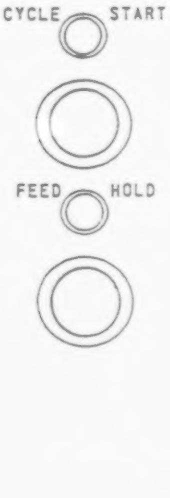

>T01;M06;_
ALTER INSERT

⑤. Depress INSERT soft key, and then the program is shifted to top side of the screen.


>T01;M06;_
ALTER INSERT
⇒
PROGRAM(MDi)
T01 ;
M06 ;
%
ALTER INSERT

⑥. Depress CYCLE START button.

Commanded program is executed.


CYCLE START
FEED HOLD

PROGRAM EDITING - ~~EXISTING~~ New PROGRAM

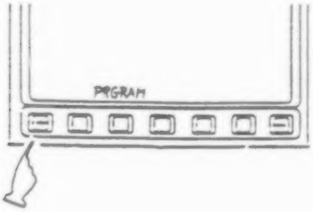
11. Program Editing

1). New program registration

①. Depress ' EDIT ' button on mode selection switches.

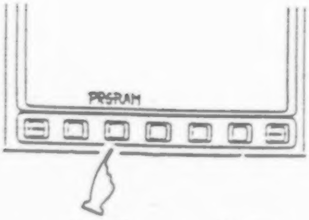
②. Display ' PRGRAM (MEMORY) ' screen.

i).



Depress Function MENU switch left below and display ' PRGRAM ' (Program).

ii).



By depressing ' PRGRAM ' soft key. ' PRGRAM (MEMORY) ' is displayed on upper left of the screen. If not displayed, repeat to depress the soft key again.

③. Key in Program No.

④. Depress ' INSERT ' soft key as shown.

If ' INSERT ' is not displayed, depress operation MENU key right below until displaying.

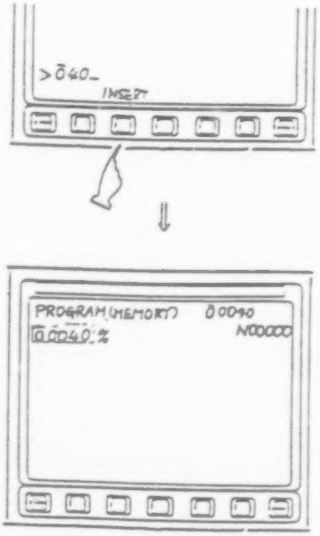
Input program No. and depress INSERT soft key. Example as follows.

Input Ō 40. If input Ō 40 ;, FORMAT ERROR is shown. Semi colon input is not required in this case.

⑤. New program No. is displayed on the screen and registered as new one.

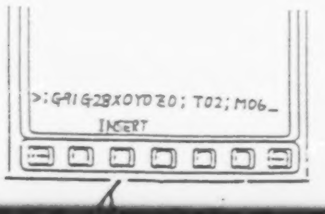
⑥. Key in the following program and depress INSERT soft key.

Max. letters in one time are 78 letters. Keeping 78 letters or within, no block number limit is given. After INSERT, each block is separated automatically.

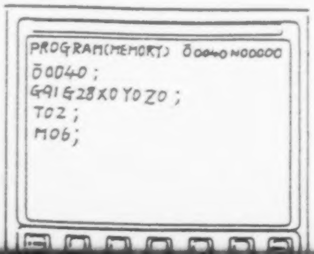


```
>Ō40_
```

```
PROGRAM(MEMORY)   Ō0040
Ō0040 %              N00000
```



```
>;G91G28X0Y0Z0;T02;M06_
```



```
PROGRAM(MEMORY) Ō0040 N00000
Ō0040;
G91G28X0Y0Z0;
T02;
M06;
```

PROGRAM EDITING - EXISTING PROGRAM

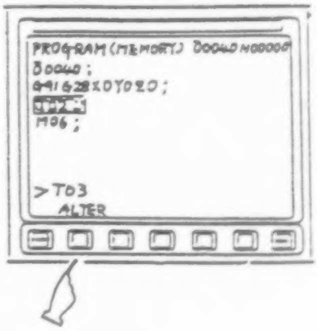
2). ALTER, INSERT and DELETE of registered program (Mode : EDIT)

①. Alteration (ALTER)

i). Move cursor to the desired position to be altered.

ii). Key in the word to be altered.

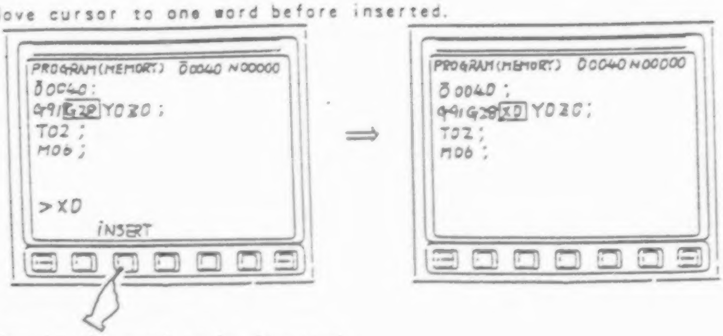
iii). Depress ' ALTER ' soft key.



```
PROGRAM(MEMORY) O0040 N00000
O0040 ;
G91G28X0Y0Z0;
[illegible]
M06;

>T03
```

②. INSERT

i). Move cursor to one word before inserted.



```
PROGRAM(MEMORY) O0040 N00000
O0040:
G91G28 Y0Z0 ;
T02 ;
M06;

>X0
```

```
PROGRAM(MEMORY) O0040 N00000
O0040 ;
G91G28X0 Y0Z0;
T02;
M06 ;
```

ii). Key in the word to be inserted.

iii). Depress ' INSERT ' soft key.

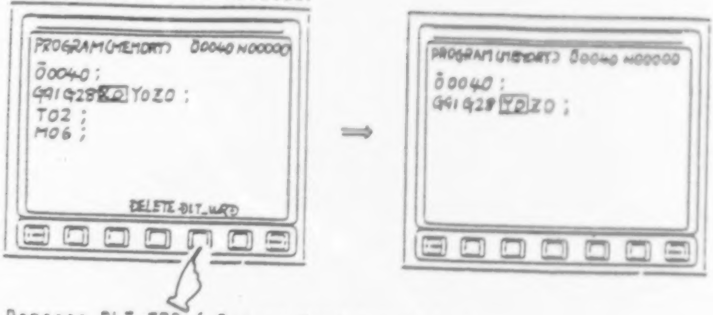
BEST COPY AVAILABLE

PROGRAM EDITING - EXISTING PROGRAM

③. DELETE

③-1. DELETE of Word (DLT-WRD)

i). Move cursor to be deleted.



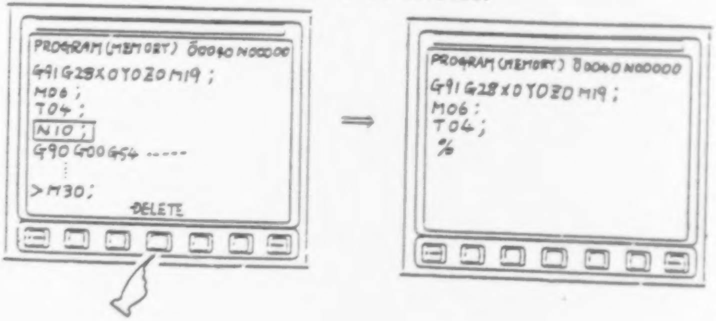
```
PROGRAM(MEMORY)   O0040 N00000
O0040 ;
G91G28X0Y0Z0 ;
T02 ;
M06 ;
```

```
PROGRAM(MEMORY) O0040 N00000
O0040 ;
G91G28Y0Z0 ;
```

ii). Depress DLT-WRD (Delete Word) soft key.

③-2. DELETE of designated range (DELETE)

i). Move cursor to the top word to be deleted.

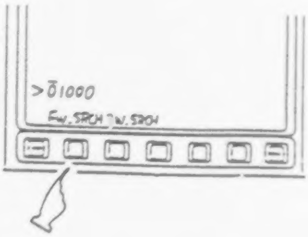


```
PROGRAM(MEMORY) O0040 N00000
G91G28X0Y0Z0M19 ;
M06 ;
T04 ;
N10 ;
G90G00G54 -----
   :
>M30;
```

```
PROGRAM(MEMORY) O0040 N00000
G91G28X0Y0Z0M19 ;
M06 ;
T04 ;
%
```

④. Key Program No. to be called and depress FW _SRCH or BW _SRCH soft key.

Keyed in Program is called.

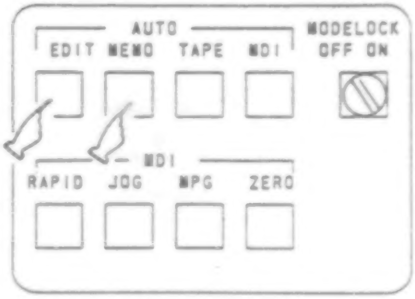


```
>O1000
```

PROGRAM CALL

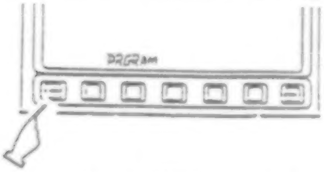
12. Program Call

①. Depress ' EDIT ' or ' MEMO ' button on mode selection switch.

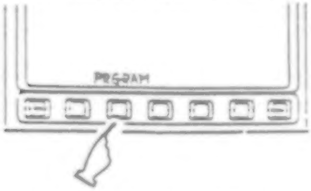

AUTO
EDIT
MEMO
TAPE
MDI
MODELOCK
OFF ON
MDI
RAPID
JOG
MPG
ZERO

②. Display PROGRAM (MEMORY) screen.

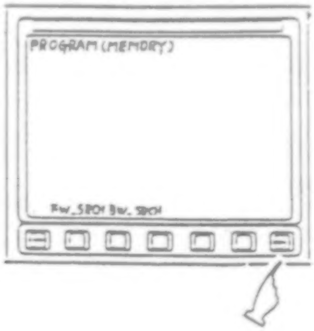
i). Depress Function MENU key left below to display ' PRGRAM ' (Program).



ii). Depressing ' PRGRAM ' soft key. PROGRAM(MEMORY) is displayed on top left of the screen. If not displayed, repeat depressing.



③. Keep depressing Right MENU key until displaying of FW_SRCH or BW_SRCH.


PROGRAM (MEMORY)
FW_SRCH BW_SRCH

13. Operation

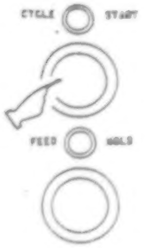
①. Depress ' MEMO ' button on mode selection switch.

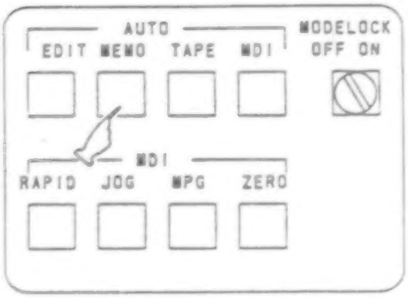
②. Check whether machining program is called or not.

③. Depress ' CYCLE START '.

⇩

Program is started automatically.


CYCLE
START
FEED
HOLD


AUTO
EDIT
MEMO
TAPE
MDI
MODELOCK
OFF ON
MDI
RAPID
JOG
MPG
ZERO

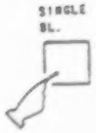
PROGRAM CHECKING METHOD

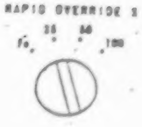
※ Program checking method :

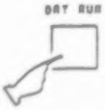
In case of new program, program mistakes might be existed somewhere in the program.

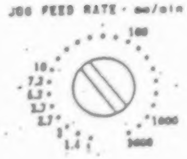
Therefore, operate the cycle with ' SIN BLOCK ' (Single Block) selection and reduce ' RAPID OVERRIDE ' to 25% without work mounting.

In case no load operation, use ' DRY RUN ' operation is recommended to shorten checking time, because all prgrammed feed rates in the program are switched to the designated feed rate by ' JOG FEED RATE ' button.




RAPID OVERRIDE %
F0
25
50
100

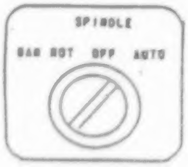



JOG FEED RATE · mm/min
100
10
7.2
5.2
3.7
2.7
2
1.4
1
1000
3600

CAUTION :

If program is not started dspite ' CYCLE START ' button is depressed, check the following items.

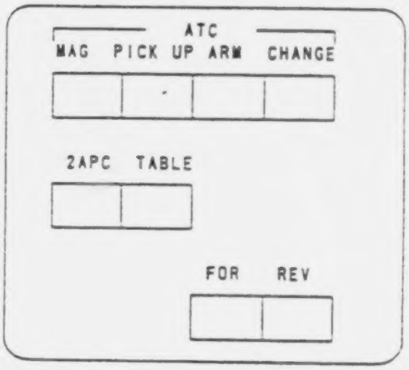
i). Check ' SPINDLE ' selector switch. If ' MAN. ROT ' is selected, CYCLE is not started. ' AUTO ' position must be selected in advance.


SPINDLE
MAN ROT
OFF
AUTO

PN - 40 MANUAL ATC OPERATION

9-2. In case of PN 40

[PN 40]



1). Manual ATC operation

①. Execute ZERO return (Reference Point Return) by MAN or MDI operation.

②. Execute spindle orientation stop and ATC position return for X-axis (2nd Reference Point Return) by MDI operation.

```
M 1 9   ;
G 9 1   G 3 0   X 0   ;
```

③. Select "MAN " side (either one of RAPID JOG. MPG or ZERO) in mode selection switch.

④. Depress ' MAG ' button and select magazine No. of new tool by depressing either FOR or REV button.

FOR : TooL magazine forward (CW)
REV : TooL magazine reverse (CCW)

BEST COPY AVAILABLE

PN - 40 MANUAL ATC OPERATION

⑤. Depress ' PICK UP ' button and keep depressing ' FOR ' button.

New tool is picked up from the magazine and moves to stand by position.

In this case, stand-by pot must be at the position as Fig.-A shown below.
When stand-by pot is positioned as Fig.-B below, ' PICK UP ' motion does not work, and DGN No.'7 5 / ATC OTHERS ' is displayed.
To recover it, select empty pot desired (if not designated, select any empty pot) in above Item-④ operation by keeping to depress ' REV ' button.

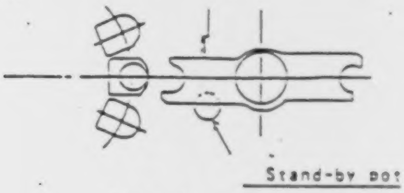


Fig.- A

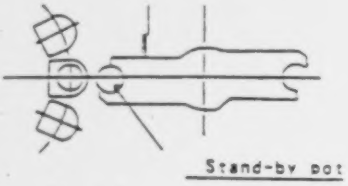


Fig.- B

⑥. Depress ' ARM ' button and keep depressing ' FOR ' button.

Swing arm is moved to spindle side, and ATC shutter is opened at the same time.

⑦. Depress ' CHANGE ' button and keep depressing ' FOR ' button.

Tool change is executed (Old Tool is picked up from the spindle and New Tool is inserted to the spindle).
By depressing ' REV ' button and keep depressing ' FOR ' button, revese motion is possible (New Tool is picked up from the spindle and Old Tool is inserted to the spindle).

[PN 40].

⑧. Return X-axis to zero position (reference point) by MDI operation.

G 9 1 G 2 8 X 0 ;.

⑨. Select ' MAN ' mode.

⑩. Select the Magazine No. for old tool to be stored in Item-④ operation.

⑪. Depress ' ARM ' button and keep depressing ' REV ' button.

Swing arm is returned to magazine side, and ATC shutter is closed.

⑫. Depress ' PICK UP ' button and keep depressing ' REV ' button.

Old Tool is stored into the magazine pot.

CAUTION : When FOR or REV button is released and turned OFF during in motion, the motion is stopped immediately.
Motion executing can be checked on MACHINE screen whether each motion is completed or not.

BEST COPY AVAILABLE

RECOVERY OF ATC

15-2. In case of PN40

1). ATC (Automatic Tool Changer) :

If EMG situation is taken place in mid-motion, basically recovery is possible by MAN.(Manual) button operation.
However, Swing Arm can not be followed to the correct position sequence (Magazine side or Spindle side) depending on mid-motion, for example :

Tool pickin up or storing motion at Tool Magazine side ;

ATC motion at spindle side ;

In this case, check EMG stop button is depressed and kept 1st, and move the swing arm manually (by hands) to either magazine side or spindle side.
Then turn ON the control power again.

1)-1. In case of EMG stop in tool changing motion :

①. NC POWER button ON ⟶ In case of power black out.

②. Control Power button ON.

After above operation, ALARM Lamp on Main Operation Panel is lit, and ' OT 007 X+ OVERTRAVEL (HARD) ' is displayed on DGN of MACHINE screen.
This alarm is appeared due to spindle orientation stop OFF by EMG stop.
Therefore, ATC recovery motion can be executed with this alarm.

③. Select MAN side (either one of RAPID, JOG, MPG or ZERO) in mode selecting switch.

④. Depress ' CHANGE ' button and keep depressing ' FOR ' button until motion end.

' REV ' button does not work in this case.

WARNING : NEVER TRY TO ROTATE SPINDLE WITHOUT A TOOL ON SPINDLE.

When machine is stopped in tool retracted position(no tool on spindle).
NEVER TRY TO ROTATE SPINDLE. because spindle orientation is OFF.
If accidentally rotated, orient the spindle to the correct position (Drive key of the spindle is parallel with X-axis : Horizontal)

If spindle orientation operation recovery is not possible or not sure, take off drive keys to avoid tool interference with the keys.

⑤. Move X-axis to minus direction away from ZERO position enough and depress RESET button ON for ALARM OFF.

⑥. Depress ' MAG ' (Magazine) button and select desired magazine (tool pot) No. by depressing either ' FOR ' or ' REV ' button.

⑦. Depress ' ARM ' button and keep depressing ' REV ' button until motion end.

Swing arm moves to magazine side and ATC shutter is closed at the same time.

BEST COPY AVAILABLE

RECOVERY OF ATC

[PN 40].

⑬. Depress ' PICK UP ' button and keep depressing ' REV ' button until motion end.

Tool is stored to tool magazine selected.

2). 2 APC

Recovery motion by MAN operation is possible for all mid-motions for 2 APC.

PN - 40 MANUAL APC OPERATION

2). Manual APC operation

①. Manual APC operation can be executed by selecting either ' MAN ' or MDI operation.

Condition :

Z-axis zero position (Reference Point) : G 9 1 G 2 8 Z 0 :
B-axis zero degree : B 0 :

②. Select ' MAN ' side (either one of RAPID JOG. MPG or ZERO) in mode selection switch.

③. Depress ' 2APC ' button and keep depressing either ' FOR ' or ' REV ' button. The following motion is executed.

Pallet UNCLAMP → Pallet index (APC shutter is indexed at the same time) → Pallet CLAMP

FOR button : For A-PALLET indexing
REV button : For B-PALLET indexing

When FOR or REV button is released and turned OFF during in motion. the motion is stopped immediately.

3). Manual NC-table CLAMP / UNCLAMP operation (In case of B-axis : NC table / OP.)

①. Select ' MAN ' side (either one of RAPID JOG. MPG or ZERO) in mode selection switch.

②. Depress ' TABLE ' button and depress either ' FOR ' for clamp or ' REV ' for unclamp.

RECOVERY OF 2APC (2 PALLET APC)

2). 2 APC (2 Pallet type Automatic Pallet Changer)

Basically, APC motion is composed of 3 motions, i.e. LOADING, INDEXING and UNLOADING.

- Unloading motion : LOADER ADVANCE ⟶ PALLET UNCLAMP ⟶ LOADER RETRACTION
- Indexing motion : A / B-PALLET INDEXING
- Loading motion : LOADER ADVANCE ⟶ PALLET CLAMP ⟶ LOADER RETRACTION

In EMG stop situation, Automatic Pallet Changing condition is reset.

Therefore, APC Loader takes NO advance motion and Retracting motion only is possible.
To resume APC motion possible, the following recovery procedure must be taken first in any case.

①. NC POWER button ON. ⟶ In case of power black out.

②. CONTROL POWER button ON.

TOOL BREAKAGE - FEED HOLD OR MID-OPERATION

14. Recovery Procedure after FEED HOLD in Mid-operation

1). Program correction in mid-operation

When program mistake is found and its correction is required in memory operation, the following procedure must be taken.

①. Depress ' SIN BLOCK ' or ' FEED HOLD ' button to stop axis feed.

②. Depress NC RESET button in MEMO (Memory) mode.

③. Return Z-axis to Zero (reference point) position in ' ZERO ' return mode. If required, return X and/or Y-axis also to Zero position.

④. Switch to ' EDIT ' mode and correct the program.

⑤. In case of restart of the cycle, move cursor to the 1st program where the tool on spindle now, and depress ' CYCLE START ' button.

WARNING :

WHEN THE CYCLE IS RESTARTED, EXECUTE WITH SINGLE BLOCK 1ST FOR SAFETY. RESTARTING POSITION IS DEPENDED ON STOPPED SITUATION IN MID-OPERATION.

CAUTION :

WHEN ' NC-RESET ' BUTTON IS DEPRESSED, INFORMATION OF CANNED CYCLE & TOOL OFFSET ARE CANCELLED.
THEREFORE, TO RESTART THE CYCLE, THESE COMMANDS MUST BE EXECUTED ONE MORE TIME IN ADVANCE.

2). Tool breakage in mid-operation

If a tool is broken during machining, operation is stopped by depressing ' FEED HOLD ' or ' EMG ' (Emergency) button.

2)-1. FEED HOLD button is depressed :

①. Stop spindle and check tool breakage and machined section.

Eliminate broken tool on the work surface out of the machine.

②. Depress NC RESET button.

③. Execute ZERO return for each axis.

④. Change tool to new one.

⑤. Tool length setting for new tool.

⑥. Move cursor to the restarting position, and depress ' CYCLE START ' button.

BEST COPY AVAILABLE

TOOL MOUNTING AND DISMOUNTING ON THE SPINDLE

7-2. Tool mounting and dismounting on the Spindle

Tool mounting and dismounting on the spindle is executed.

①. Tool dismounting from spindle

i). Select ' UNCLAMP ' side of ' SPINDLE COLLET CLAMP / UNCLAMP ' switch.

ii). Hold the tool and holder manually to prevent tool dropping.

iii). Depress SPINDLE COLLET button at center side.

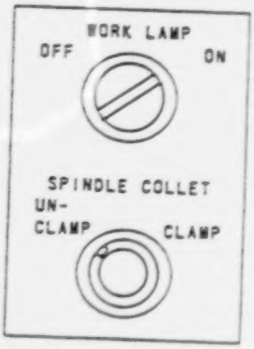
CAUTION : THIS OPERATION IS NOT POSSIBLE IN HN-B SERIES IF M19 (SPINDLE ORIENTATION STOP) IS NOT EXECUTED YET.

②. Tool mounting to Spindle

i). Select ' CLAMP ' side of ' SPINDLE COLLET CLAMP / UNCLAMP ' switch.

ii). Insert the tool and holder together to the spindle manually.

iii). Depress SPINDLE COLLET button at center side.

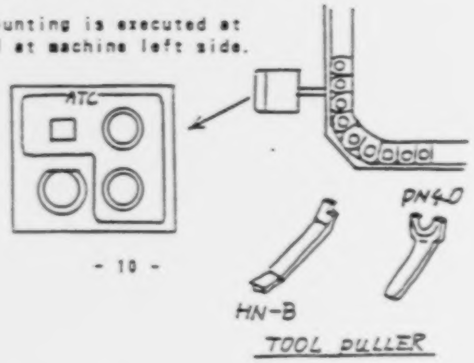


TOOL MOUNTING AND DISMOUNTING AT TOOL MAGAZINE

7. Tool mounting and dismounting

7-1. Tool mounting and dismounting at Tool Magazine

Tool mounting & dismounting is executed at Tool Magazine located at machine left side.



①. Depress ' MAN. OPER. ' switch of ATC Operation Box.

→ MAN. OPER. lamp is lit in Green.

②. Index a desired tool pot either CW or CCW direction.

③. When tool is dismounted, use Tool Dismounting kit shown in previous page (P.10). When mounting, just insert each tool and holder set the tool pot.

④. After above operation finished, depress MAN. OPER. switch again. Green lamp is turned OFF.

CAUTION : IF THIS OPERATION IS NOT EXECUTED, ATC MOTION BY MEMORY OR MDI OPERATION CAN NOT BE EXECUTED.

BEST COPY AVAILABLE

MACHINE SCREEN DISPLAY

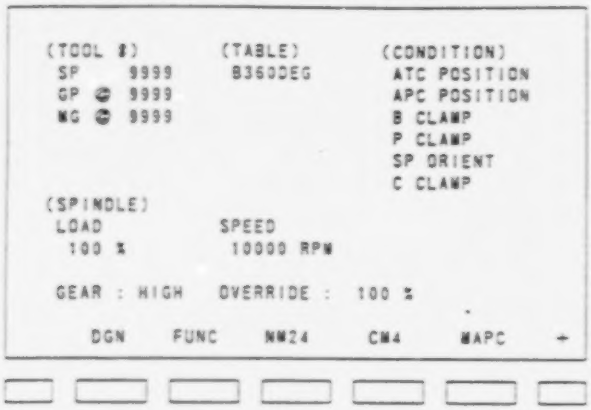
16. MACHINE Screen

16-1. Display operation of MACHINE screen

①. Depress ' MC DISPLAY ' button (MC DISPLAY button is lit with green).

MACHINE screen is displayed on CRT screen.

SCREEN 1.1 (MACHINE SCREEN, MAIN)



```
(TOOL #)         (TABLE)         (CONDITION)
 SP     9999      B360DEG         ATC POSITION
 GP ●  9999                       APC POSITION
 MG ●  9999                       B CLAMP
                                  P CLAMP
                                  SP ORIENT
                                  C CLAMP
(SPINDLE)
 LOAD            SPEED
  100 %           10000 RPM

 GEAR : HIGH   OVERRIDE :  100 %

    DGN    FUNC    NM24    CM4    MAPC    +
```

(TOOL #)

SP : Tool No. on Spindle
GP ● : Tool No. on Gripper
MG ● : Tool No. of Magazine indexed

● : Tool ON

(TABLE)

B360DEG : Table indexed position

(SPINDLE)

LOAD : Spindle load meter (OP.)
100 %

SPEED : Spindle speed meter
10000 RPM (OP.)

GEAR : HIGH -- Spindle gear

HIGH : High range
LOW : Low range

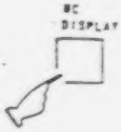
OVERRIDE : 100 %

Spindle override

(CONDITION) : Machine condition is displayed.

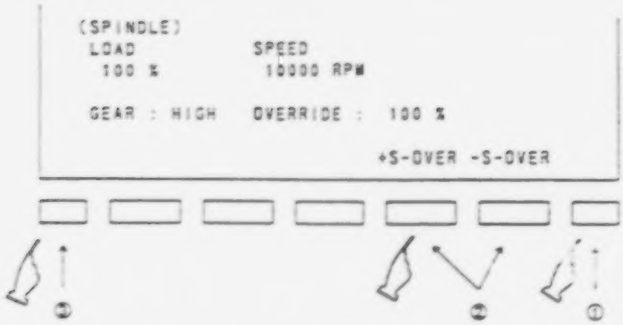
ATC POSITION : ATC tool change position
APC POSITION : APC pallet change position
B CLAMP : B-axis clamped
P CLAMP : Pallet clamped
SP ORIENT : Spindle orientation completed
C CLAMP : C-axis clamped (OP.)

②. Depress ' MC DISPLAY ' button again to return to NC screen.



MACHINE SCREEN DISPLAY

16-2. Spindle Override Setting (OP.)



```
(SPINDLE)
 LOAD           SPEED
  100 %          10000 RPM

 GEAR : HIGH   OVERRIDE :  100 %

                             +S-OVER -S-OVER
```

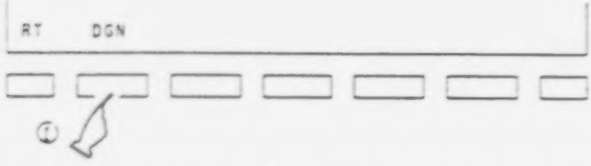
①. Depress MENU key.

②. Set override by + S or − S.

③. Setting finished

16-3. DGN (Diagnosis) DISPLAY screen

①. Depress soft key ' DGN '.

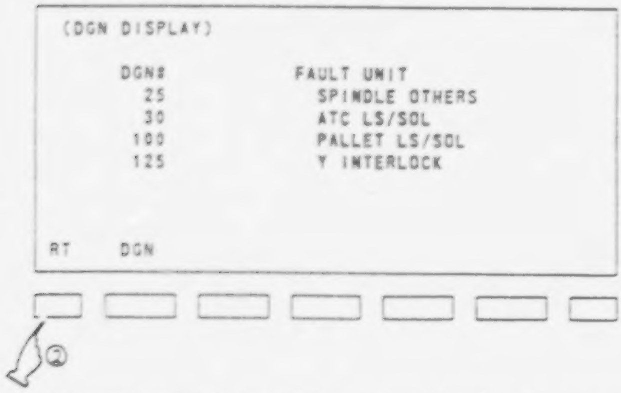


```
RT    DGN
```

RT : Return

In alarm situation, DGN No. and alarm unit are displayed on the screen.

SCREEN 2 (DGN DISPLAY)



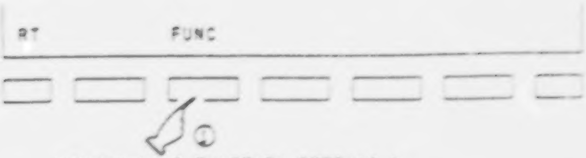
```
(DGN DISPLAY)

     DGN#          FAULT UNIT
       25            SPINDLE OTHERS
       30            ATC LS/SOL
      100            PALLET LS/SOL
      125            Y INTERLOCK



RT    DGN
```

②. Depressing MENU key left, screen display is returned to SCREEN 1.1 (P.32)

MACHINE SCREEN DISPLAY

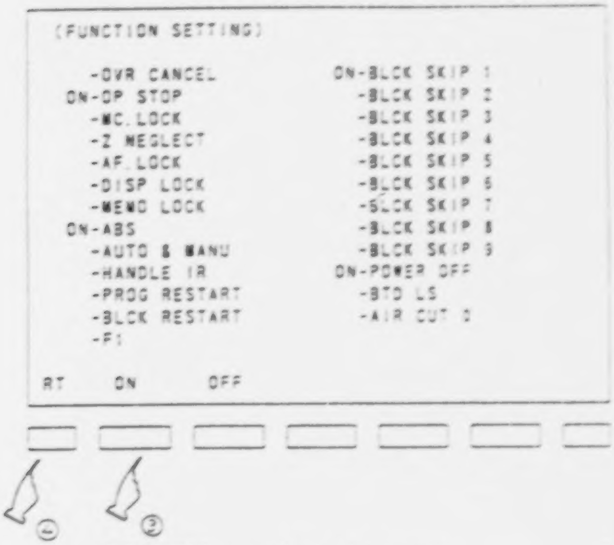
15-4. FUNC (Function) switch setting

①. Depress ' FUNC ' soft key.



```
RT            FUNC
```

SCREEN 3 (FUNCTION SETTING)



```
(FUNCTION SETTING)

   -OVR CANCEL              ON-BLCK SKIP 1
 ON-OP STOP                    -BLCK SKIP 2
   -MC.LOCK                    -BLCK SKIP 3
   -Z NEGLECT                  -BLCK SKIP 4
   -AF.LOCK                    -BLCK SKIP 5
   -DISP LOCK                  -BLCK SKIP 6
   -MEMO LOCK                  -BLCK SKIP 7
 ON-ABS                        -BLCK SKIP 8
   -AUTO & MANU                -BLCK SKIP 9
   -HANDLE IR               ON-POWER OFF
   -PROG RESTART               -BTD LS
   -BLCK RESTART               -AIR CUT 0
   -F1

RT     ON       OFF
```

All function including optional are displayed in this screen.

Refer to FANUC OPERATOR'S MANUAL and NIIGATA INSTRUCTION MANUAL in details for each function.

<u>CAUTION</u> : ABS function must be kept ON all the time.

OVR CANCEL : Override cancel
OP STOP : Optional stop
MC.LOCK : Machine lock
Z NEGLECT : Z-axis neglection
AF.LOCK : Additional function lock
DISP LOCK : Display lock
MEMO LOCK : Memory lock
ABS : Absolute
AUTO & MANU : Automatic & Manual
HANDLE IR : Handle interruption
PROG RESTART : Program restart
BLCK RESTART : Block restart
F1 : F-1 (one) digit feed

BLCK SKIP : Block skip
BTD LS : Limit switch type Broken Tool Detection (OP.)
AIR CUT 0 : Air cut zero function(OP.)

②. Move cursor to the desired item to be set using PAGE ↑ ↓ CURSOR ← → keys.

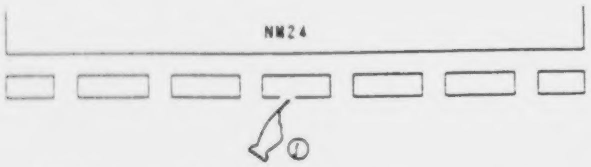
③. Depress ' ON ' or ' OFF ' soft key for ON/OFF setting.

④. Depress ' RT ' key to return to original MACHINE screen.

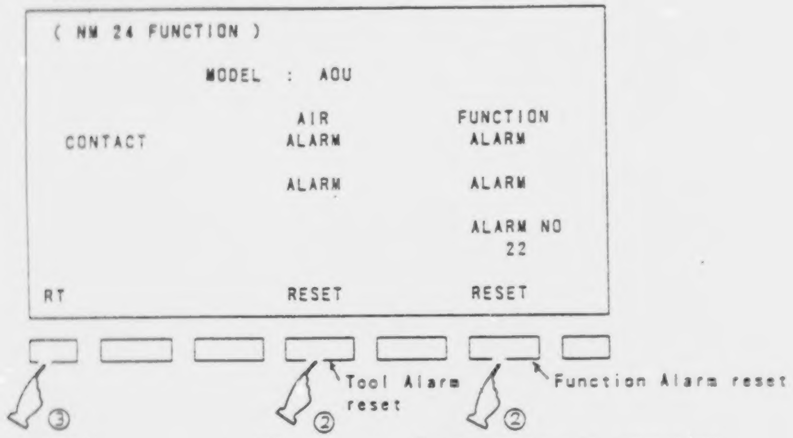
MACHINE SCREEN DISPLAY

16-5. NM 24 Function screen (OP.) : AOU, AMU, ATM etc.

①. Depress ' NM24 ' soft key.



SCREEN 4 (NM24 FUNCTION)



```
( NM 24 FUNCTION )

             MODEL  :  AOU

                      AIR              FUNCTION
 CONTACT              ALARM             ALARM

                      ALARM             ALARM

                                        ALARM NO
                                           22

RT                    RESET             RESET
```

MODEL : AOU ------ Function name in execution

CONTACT : When Dection Signal of Monitoring Probe is output, this signal is turned ON.

AIR ALARM : In alarm situation in Air pressure, ALARM signal is displayed.

FUNCTION ALARM : In alarm situation of a function, ALARM signal is displayed.

ALARM NO : Alarm No. is displayed.

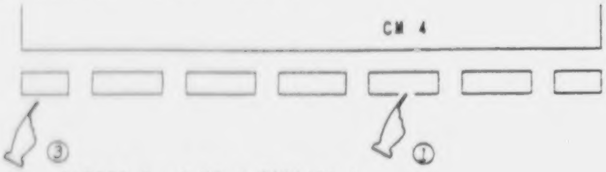
②. Depress either AIR ALARM or FUNCTION ALARM ' RESET ' button.

③. Depress ' RT ' key to return to original MACHINE screen.

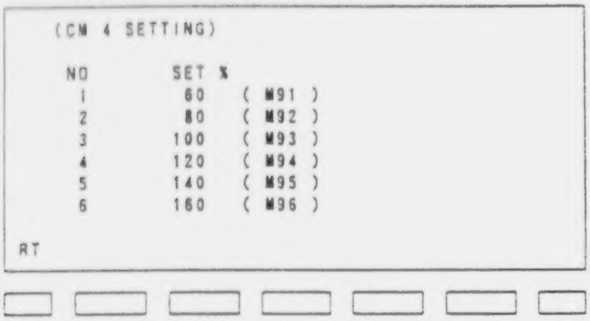
BEST COPY AVAILABLE

16-6. CM 4 Function setting screen

①. Depress ' NM24 ' soft key.



SCREEN 5 (CM 4 SETTING)



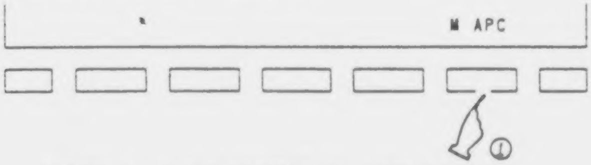
```
(CM 4 SETTING)

 NO        SET %
  1          60   ( M91 )
  2          80   ( M92 )
  3         100   ( M93 )
  4         120   ( M94 )
  5         140   ( M95 )
  6         160   ( M96 )

RT
```

②. Move cursor to the desired item and input set value (%).

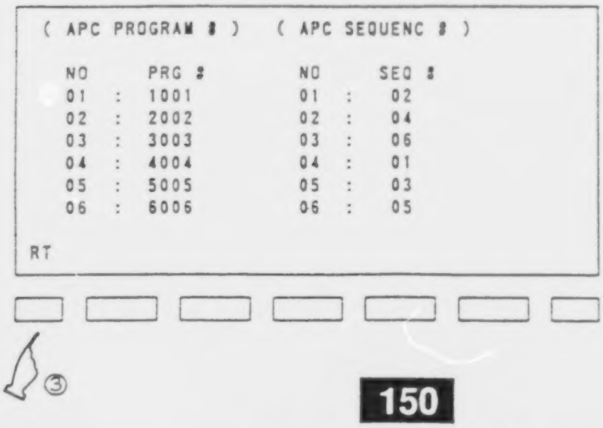
③. Depress ' RT ' key to return to original MACHINE screen.

16-7. Program No. setting screen of APC (OP.)

①. Depress ' M APC ' soft key.



SCREEN 6 (APC PROGRAM NO. SETTING)

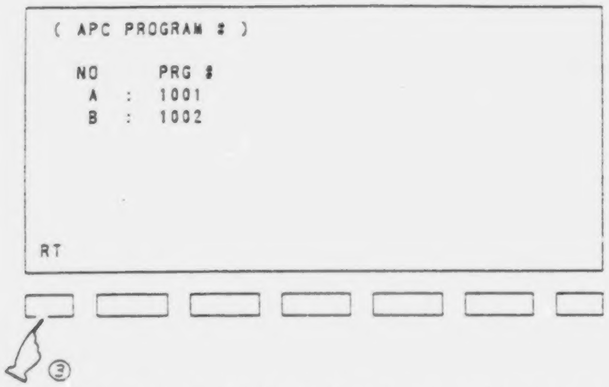


```
( APC PROGRAM # )   ( APC SEQUENC # )

  NO      PRG #         NO      SEQ #
  01  :   1001          01  :    02
  02  :   2002          02  :    04
  03  :   3003          03  :    06
  04  :   4004          04  :    01
  05  :   5005          05  :    03
  06  :   6006          06  :    05

RT
```

PRG # : Program No.
SEQ # : Pallet change sequence

【 In case of 6 APC 】

MACHINE SCREEN DISPLAY



```
( APC PROGRAM # )

  NO      PRG #
   A  :   1001
   B  :   1002



RT
```

[In case of 2 APC]

②. Move cursor to the desired position and input set value by numeric keys.

③. Depress ' RT ' key to return to original MACHINE screen.

Note-1 : Refer to ' FANUC Series 15-MA OPERATOR'S MANUAL (B-61224E) ' for more details on MACHINE screen.

Note-2 : Items with (OP.) means optional specifications. Refer to each machine specification.

BEST COPY AVAILABLE

NIIGATA CNC HMC

ALARM MESSAGE DISPLAY - (example)

§ 9 TROUBLE DIAGNOSIS LIST (DGN TABLE) PN 40

(STD.)

DGN No	D G N	Contents	Normal
1	Spindle alarm	Spindle stop despite of spindle rotation command ON. (Reset: Select MAN mode.)	R44.2 OFF
2	Spindle rotation confirmation OFF in spindle drive unit	Spindle drive unit signal confirmation error	X6.0 ON
3	Spindle stop confirmation OFF in spindle drive unit	Spindle drive unit signal confirmation error	X6.1 ON
4	Spindle rotation confirmation ON at spindle drive unit	Spindle drive unit signal return error	X6.0 OFF
5	Spindle stop confirmation ON at spindle drive unit	Spindle drive unit signal return error	X6.1 OFF
6	Spindle orientation confirmation OFF at spindle drive	Spindle drive unit signal return error	X6.3 ON
9	Spindle high speed range LS OFF (LS-16)	Stroke end LS confirmation error	LS-16 X3.0 ON
10	Spindle low speed range LS OFF (LS-15)	Stroke end LS confirmation error	LS-15 X3.1 ON
11	Spindle clamped LS OFF (LS-13)	Stroke end LS confirmation error	LS-13 X3.3 ON
12	Spindle unclamped LS OFF (LS-14)	Stroke end LS confirmation error	LS-14 X3.2 ON
13	Spindle high range LS ON (LS-16)	Limit switch return error	LS- 16 X3.0 OFF
14	Spindle low range LS ON (LS-15)	Limit switch return error	LS-15 X3.1 OFF
15	Spindle clamped LS ON (LS-13)	Limit switch return error	LS-13 X3.3 OFF
16	Spindle unclamped LS ON (LS-14)	Limit switch return error	LS-14 X3.2 OFF
19	Spindle high range SOL OFF (SOL-309B)	Solenoid motion error (Continues to next.)	Y1.4 SOL-309B ON

NIIGATA ENGINEERING CO., LTD. T-902

NIIGATA CNC HMC

ALARM MESSAGE DISPLAY -(Example)

§ 9 TROUBLE DIAGNOSIS LIST (DGN TABLE) PN 40

(STD.) Revised on Aug. 10. 1990

DGN No	D G N	Contents	Normal
74	ATC pos. confirmation OFF	X-axis 2nd reference point (G30) ┐ X-axis reference point (G28) ┘ Either one of above is OFF.	R1.6 ON
75	Tool pick up refused	Tool pick up is prohibited. (Reset: Execute tool storing motion manually once.)	K2.7 ON
76	Arm swing refused to go to magazine side	Tool in magazine & tool in gripper	R56.7 ON
78	Magazine index command 0	Magazine indexes with 0 (zero) command. (Recommand by T-code or reset spindle tool number.)	R426. R427 ≠ 0
79	Larger command for magazine index	Larger No. by T-command is given than magazine tool number	R426. R427 ≦D46. D47
80	Magazine stop position discrepancy	Stop position is not coincided with commanded position	R62.5 OFF
81	B-axis drive unit ubnormal (72T. 360T)	B-axis drive unit ubnormal (Misposition, vibration, command data ≧ 360° & etc.) FEED HOLD comes. [Reset : After depressing RESET SW (SW-1), turn ON control power.]	X4.4 OFF
82	B-axis drive unit servo alarm (72T. 360T)	B-axis drive unit servo alarm (TG alarm, low voltage, overload, overcurrent, overspeed & etc. : Refer to MANUAL.) EMERGENCY STOP comes. [Reset: After eliminating the cause of alarm, depress RESET SW (SW-1) on B-axis drive unit, then turn ON control power.]	X4.5 ON
84	B-axis clamped LS OFF [72T, 360T LS-17 NC-T: PS-2]	Limit switch confirmation error (72T, 360T) LS-17 ┐ ├ ON X3.6 ┘	(NC-T) PS-2 ┐ ├ ON X3.7 ┘
85	B-axis unclamped LS OFF (72T. 360T) (LS-18)	Limit switch confirmation error (To be continued.)	LS-18 X3.5 ON

Niigata CNC Horizontal M.C.

Restart Program (Safe Home)

PN40 Machine

```
07998
G91 G28 Z0 M19;
G91 G28 X0 Y0 M9;
G90 G17 G20 G40 G49 G80;
#3003 = 0;
#3004 = 0;
#508 - 0;
M99;
90
```

07998 is a sub-program and will normally be imbedded at or near the top of your main program.

Above variables explained.

#3003 = 0	Means turn single block off
#3004 = 0	Means turn feed hold to off
#508 = 0	Means this common variable is holding

Your Niigata Notes

Niigata CNC Horizontal M.C.

Questions: Machine & Control

1. What is the maximum tool mass allowable on the automatic tool changer?

   A] ____________________

2. How accurately can the machine position in X, Y, and Z?

   A] ____________________

3. A tool is calculated by the operator to run at 11,000 RPM on this machine. Will the machine reach this high RPM?

   A] ____________________

   Because max. RPM is____________________

4. Which two "G" codes are acceptable for tool length compensation?

   A] ____________________

   B] ____________________

Questions: Machine & Control

1. Can this machine do circulator interpolation using multi-quadrant circular interpolation?
   A] ____________________

2. Explain the advantage of multi-quadrant circular interpolation.

____

____

____

3. Which work coordinate offsets ("G" codes) are allowable on this machine?
   A] ____________________
   B] ____________________
   C] ____________________
   D] ____________________
   E] ____________________
   F] ____________________

4. How many sub-programs can be contained in a main program?
   A] ____________________
   B] ____________________

Your Notes:

Section Eleven

Your Notes:

Training Daily Guide

Session Eleven

During this two hour segment, participants will:

1. Define machinability factors of:
   Speed, feed and depth of cut
2. Understand the use of speeds and feed conversion charts
3. Determine correct cutting speeds and feeds for common cutting tools used in milling applications
4. Understand terms:
   Cutting speed and surface feet per minute
5. Understand basic formulas used for determining RPM and feed rate
6. Determine correct feed rates for rigid and non-rigid tapping applications

Speeds and Feeds

The cutting conditions that determine the rate of metal removal are:

The cutting speed (CS),
The feed rate, and
The depth of cut.

The cutting speed is a value used to determine RPM (Speed).

The feed rate variables are the number of teeth, feed per tool, and RPM.

The depth of cut will be limited by the amount of metal that is to be machined, by the power available` on the machine tool, as well as the setup.

Speeds and Feeds

Tool life is influenced most by:

1. Cutting speed (RPM)
2. Feed rate
3. Depth of cut

The first step in selecting cutting conditions is to select: ____________________

The second step in is to select:

____

The third step in is to select:

____

UNIT: SPEEDS AND FEEDS - RPM VERSUS CUTTING SPEED
CONVERSION CHART

DIAMETER in Inches	PERIPHERAL SPEED IN FEET PER MINUTE																		
	10	20	30	40	50	60	70	80	90	100	120	140	160	180	200	225	250	275	300
	REVOLUTIONS PER MINUTE																		
1/8	306	611	917	1222	1528	1834	2139	2445	2750	3056	3667	4278	4890	5500	6115	6876	7639	8404	9167
1/4	153	306	458	611	764	917	1070	1222	1376	1528	1734	2139	2445	2750	3056	3438	3820	4202	4584
3/8	102	204	306	408	509	611	713	815	916	1018	1222	1425	1629	1832	2036	2292	2546	2801	3056
1/2	76	153	229	306	382	458	535	611	688	764	917	1070	1222	1375	1528	1719	1910	2101	2292
5/8	61	122	184	245	306	367	428	489	552	611	733	857	979	1102	1224	1375	1528	1681	1833
3/4	51	102	153	203	254	306	357	408	458	508	611	711	813	914	1016	1146	1273	1400	1528
7/8	44	88	131	175	219	262	306	349	392	438	526	611	701	788	876	982	1091	1200	1310
1	38	76	115	153	191	229	267	306	344	382	458	535	611	688	764	859	954	1050	1146
1-1/4	31	62	92	123	153	183	214	245	274	306	367	428	490	551	611	687	764	840	917
1-1/2	25	51	76	102	127	153	178	204	230	254	306	356	406	457	508	573	636	700	764
1-3/4	22	44	65	87	109	131	153	175	196	218	262	306	349	392	436	491	546	600	655
2	19	38	57	76	95	115	134	153	172	191	229	267	306	344	382	429	477	525	573
2-1/4	17	34	51	68	85	102	119	136	153	170	204	238	272	306	340	382	424	467	509
2-1/2	15	30	46	61	76	92	107	122	138	153	184	213	245	275	306	344	382	420	458
2-3/4	14	28	42	56	70	83	97	111	125	139	167	195	222	250	278	312	347	382	417
3	13	26	38	51	64	76	89	102	114	127	153	178	203	228	254	286	318	350	382
3-1/4	12	24	35	47	58	70	82	94	105	117	140	164	188	211	234	264	294	323	352
3-1/2	11	22	33	44	55	66	76	87	98	109	131	153	174	196	218	245	273	300	327
3-3/4	10	20	31	41	51	61	71	82	92	102	122	143	163	184	205	229	255	280	305
4	9	19	29	38	48	57	67	76	86	96	115	134	153	172	191	215	239	263	286
4-1/4	9	18	27	36	45	54	63	72	81	90	108	126	144	162	180	202	225	247	270
4-1/2	8	17	25	34	42	51	59	68	76	85	102	119	136	153	170	191	212	233	255
4-3/4	8	16	24	32	40	48	56	64	72	80	96	112	129	145	161	180	201	221	241
5	7	15	23	31	38	46	54	61	69	76	92	107	122	138	153	171	191	210	229
5-1/4	7	15	22	29	36	44	51	58	65	73	87	102	116	131	145	163	181	199	218
5-1/2	7	14	21	28	35	42	49	56	62	69	83	97	111	125	139	156	173	190	208
5-3/4	6	13	20	27	33	40	46	53	60	66	80	93	106	120	133	149	166	182	199
6	6	13	19	25	32	38	45	51	57	64	76	89	102	114	127	143	159	174	190

BEST COPY AVAILABLE

Speeds and Feeds

UNIT: SPEEDS AND FEEDS - RPM VERSUS CUTTING SPEED
CONVERSION CHART

DIAMETER in Inches	PERIPHERAL SPEED IN FEET PER MINUTE																		
	350	400	450	500	550	600	650	700	750	800	850	900	950	1000	1050	1100	1150	1200	1250
	REVOLUTIONS PER MINUTE																		
1/8	10696	12224	13752	15279	16807	18335	19862	21390	22918	24448	25974	27502	29030	30560	32088	33613	35141	36669	38197
1/4	5348	6112	6876	7639	8403	9168	9931	10696	11460	12224	12987	13751	14515	15280	16044	16807	17571	18335	19099
3/8	3566	4076	4585	5095	5622	6112	6621	7130	7639	8149	8658	9167	9677	10186	10695	11204	11714	12223	12732
1/2	2674	3057	3439	3821	4202	4584	4966	5348	5730	6112	6493	6875	7260	7639	8021	8403	8785	9167	9549
5/8	2139	2445	2751	3057	3362	3667	3972	4278	4584	4890	5194	5500	5806	6112	6418	6723	7028	7334	7639
3/4	1783	2038	2292	2547	2801	3056	3310	3565	3820	4075	4329	4584	4838	5093	5348	5602	5857	6112	6366
7/8	1528	1746	1965	2183	2401	2620	2837	3056	3274	3492	3711	3929	4147	4365	4584	4802	5020	5238	5457
1	1337	1528	1719	1910	2101	2292	2483	2674	2865	3055	3247	3438	3629	3820	4012	4202	4393	4584	4775
1-1/4	1069	1222	1375	1528	1681	1834	1986	2139	2292	2445	2597	2750	2903	3056	3208	3361	3514	3667	3820
1-1/2	891	1018	1146	1273	1400	1528	1655	1783	1909	2037	2165	2292	2419	2546	2674	2801	2938	3056	3184
1-3/4	764	873	982	1091	1200	1310	1419	1528	1637	1746	1855	1964	2074	2183	2292	2401	2510	2619	2728
2	668	764	859	955	1050	1146	1241	1337	1432	1528	1622	1719	1814	1910	2005	2101	2196	2292	2387
2-1/4	594	679	764	849	934	1018	1103	1188	1273	1358	1443	1528	1613	1698	1782	1867	1956	2037	2122
2-1/2	534	611	687	764	840	916	993	1069	1146	1222	1299	1375	1451	1528	1604	1681	1758	1833	1910
2-3/4	486	555	625	694	764	833	903	972	1041	1110	1181	1250	1320	1389	1458	1528	1598	1667	1736
3	445	509	572	637	700	764	828	891	954	1018	1082	1146	1210	1273	1336	1401	1464	1528	1592
3-1/4	411	470	528	587	646	704	764	823	882	940	999	1058	1117	1175	1234	1293	1352	1411	1468
3-1/2	381	436	490	546	600	654	709	764	819	873	928	982	1037	1091	1144	1201	1256	1310	1364
3-3/4	356	407	458	509	560	610	662	713	764	815	866	917	968	1019	1070	1121	1172	1223	1274
4	334	382	429	478	525	572	621	668	715	764	812	859	907	955	1002	1050	1098	1145	1194
4-1/4	314	359	404	449	494	539	584	629	674	718	764	809	854	899	942	987	1034	1079	1123
4-1/2	297	339	382	424	467	509	552	594	636	679	721	764	806	849	891	932	976	1017	1061
4-3/4	281	321	361	402	442	482	523	563	603	643	682	724	764	804	844	885	924	965	1006
5	267	305	343	382	420	458	496	535	572	611	649	688	726	764	802	840	878	916	954
5-1/4	255	290	327	364	402	436	473	510	544	582	619	656	691	728	764	800	837	873	909
5-1/2	242	277	312	347	382	416	451	486	520	556	590	625	660	694	730	764	799	833	868
5-3/4	232	266	298	332	364	398	431	464	498	532	564	596	630	664	696	731	762	796	830
6	222	254	286	318	350	380	414	446	476	509	540	573	605	637	668	701	732	764	796

BEST COPY AVAILABLE

Speeds and Feeds

UNIT: SPEEDS AND FEEDS - CONVERSION CHART FOR FEED

	INCHES PER TOOTH TO INCHES PER MINUTE														
NUMBER in Teeth	FEED PER TOOTH														
	.0002	.0004	.0006	.0008	.001	.002	.004	.006	.008	.010	.012	.014	.016	.018	.020
	FEED (INCHES PER MINUTE) FOR ONE RPM														
2	.0004	.0008	.0012	.0016	.002	.004	.008	.012	.016	.020	.024	.028	.032	.036	.040
3	.0006	.0012	.0018	.0024	.003	.006	.012	.018	.024	.030	.040	.042	.048	.054	.060
4	.0008	.0016	.0024	.0032	.004	.008	.016	.024	.032	.040	.048	.056	.064	.072	.080
6	.0012	.0024	.0036	.0048	.006	.012	.024	.036	.048	.060	.072	.084	.096	.108	.120
8	.0016	.0032	.0048	.0064	.008	.016	.032	.048	.064	.080	.096	.112	.128	.144	.160
10	.0020	.0040	.0060	.0080	.010	.020	.040	.060	.080	.100	.120	.140	.160	.180	.200
12	.0024	.0048	.0072	.0096	.012	.024	.048	.072	.096	.120	.144	.168	.192	.216	.240
14	.0028	.0056	.0084	.0112	.014	.028	.056	.084	.112	.140	.168	.196	.224	.252	.280
16	.0032	.0064	.0096	.0128	.016	.032	.064	.096	.128	.160	.192	.224	.256	.288	.320
18	.0036	.0072	.0108	.0144	.018	.036	.072	.108	.144	.180	.216	.252	.288	.324	.360
20	.0040	.0080	.0120	.0160	.020	.040	.080	.120	.160	.200	.240	.280	.320	.360	.400
22	.0044	.0088	.0132	.0176	.022	.044	.088	.132	.176	.220	.264	.308	.352	.396	.440
24	.0048	.0096	.0144	.0192	.024	.048	.096	.144	.192	.240	.288	.336	.384	.432	.480
26	.0052	.0104	.0156	.0208	.026	.052	.104	.156	.208	.260	.312	.364	.416	.468	.520
28	.0056	.0112	.0168	.0224	.028	.056	.112	.168	.224	.280	.336	.392	.448	.504	.560
30	.0060	.0120	.0180	.0240	.030	.060	.120	.180	.240	.300	.360	.420	.480	.540	.600
32	.0064	.0128	.0192	.0256	.032	.064	.128	.192	.256	.320	.384	.448	.512	.576	.640
36	.0072	.0144	.0216	.0288	.036	.072	.144	.216	.288	.360	.432	.504	.576	.648	.720
40	.0080	.0160	.0240	.0320	.040	.080	.160	.240	.320	.400	.480	.560	.640	.720	.800
44	.0088	.0176	.0264	.0352	.044	.088	.176	.264	.352	.440	.528	.616	.704	.792	.880
48	.0096	.0192	.0288	.0384	.048	.096	.192	.288	.384	.480	.576	.672	.768	.864	.960
52	.0104	.0208	.0312	.0416	.052	.104	.208	.312	.416	.520	.624	.728	.832	.936	1.040
56	.0112	.0224	.0336	.0448	.056	.112	.224	.336	.448	.560	.672	.784	.896	1.008	1.120
60	.0120	.0240	.0360	.0480	.060	.120	.240	.360	.480	.600	.720	.840	.960	1.080	1.200
72	.0144	.0288	.0432	.0576	.072	.144	.288	.432	.576	.720	.864	1.008	1.152	1.296	1.440
90	.0180	.0360	.0540	.0720	.090	.180	.360	.540	.720	.900	1.080	1.260	1.440	1.620	1.800

BEST COPY AVAILABLE

EXAMPLE: FOR .006 INCH FEED PER TOOTH, USING A 20 TOOTH CUTTER, TURNING AT 78 RPM.

FEED FOR ONE RPM (FROM TABLE) = .120
FEED FOR 78 RPM = .120 * 78 = 9.39 INCHES PER MINUTE

Speeds and Feeds

UNIT: SPEEDS AND FEEDS - CENTER DRILLING

SUGGESTED CENTER DRILLING SPEEDS IN R.P.M.

SIZE	SUGGESTED RPM	SUGGESTED FEED (I.P.R.)
0	2,000	.001 - .003
1	1,750	.001 - .003
2	1,500	.002 - .006
3	1,300	.002 - .006
4	1,100	.002 - .006
5	900	.002 - .006
6	700	.002 - .006
7	500	.003 - .008

I.P.M. = I.P.R. * RPM

Speeds and Feeds

UNIT: SPEEDS AND FEEDS - SPOT DRILLING

SUGGESTED SPOT DRILLING SPEEDS IN R.P.M.

FORMULA

$$\text{RPM} = \frac{\text{S.F.P.M.} * 4}{\text{DIA.}} * 2$$

MATERIAL	SPOT DRILL SPEED S.F.P.M.
ALUMINUM	200
BRASS	200
MAGNESIUM	150
CAST IRON	70
LOW CARBON STEEL	80
STAINLESS STEEL	30
THERMOPLASTICS	100
TITANIUM	20

SUGGESTED FEEDS FOR HIGH SPEED STEEL SPOT DRILLS

SPOT DRILL DIA.	FEED / REV.
1/8	.001
1/4	.002
3/8	.003
1/2	.0035
3/4	.005
1.0	.007

Speeds and Feeds

UNIT: SPEEDS AND FEEDS - DRILLING

SUGGESTED DPEEDS FOR HIGH SPEED STEEL DRILLS

FORMULA $$RPM = \frac{S.F.P.M. * 4}{DIA.}$$

MATERIAL	REGULAR DRILLS SPEED SFPM	TURBOFLUTE DRILLS SPEED SFPM
ALUMINUM	200	200
BRASS	200	200
MAGNESIUM	150	200
CAST IRON	70	70
LOW CARBON STEEL	80	80
STAINLESS STEEL	30	30
THERMOPLASTICS	100	100
TITANIUM	20	20

SUGGESTED FEEDS FOR HIGH SPEED STEEL DRILLS

DRILL DIA.	REGULAR DRILLS FEED / REV	TURBO-FLUTE DRILLS FEED / REV
UNDER 1/8	.001 - .002	.001 - .002
1/8 - 1/4	.002 - .003	.003 - .008
1/4 - 1/2	.004 - .007	.006 - .014
1/2 - 1.0	.007 - .015	.010 - .030
1/0 +	.015 - .025	.022 - .050

I.P.M. = I.P.R. * RPM

I.P.M. = INCH PER MINUTE

QUICK FORMULA

RECOMMENDED I.P.R

65 | DRILL DIA.

UNIT: SPEEDS AND FEEDS - REAMING

SUGGESTED SPEEDS FOR HIGH SPEED STEEL REAMERS

FORMULA $$RPM = \frac{S.F.P.M. * 4}{DIA.}$$

MATERIAL	CUTTING SPEED S.F.P.M.
ALUMINUM	100
BRASS	100
MAGNESIUM	75
CAST IRON	50
LOW CARBON STEEL	60
STAINLESS STEEL	20
THERMOPLASTICS	50
TITANIUM	20

SUGGESTED FEEDS FOR HIGH SPEED STEEL REAMERS

REAMER DIAMETER	FEED PER REVOLUTION I.P.R.
UNDER 1/8 DIA.	.002 - .004
1/8 - 1/4	.004 - .006
1/4 - 1/2	.006 - .008
1/2 - 1.0	.008 - .015
1.0 +	.015 - .025

FEED RATE FORMULA

I.P.M. = I.P.R. * RPM

I.P.M. = INCH PER MINUTE

UNIT: SPEEDS AND FEEDS - COUNTERSINKING

SUGGESTED COUNTERSINKING SPEEDS IN R.P.M.

C'SINK SIZE	ALUMINUM	C. IRON	STEEL
.25 DIA	500	200	175
.50 DIA	250	150	150
.75 DIA	200	100	125
1.0 DIA	150	75	80
2.0 DIA	100	75	75

FEED RATE RANGE - .003 TO .010 I.P.R.

NOTE: FEED RATE VARIES FROM MACHINE TO MACHINE.

I.P.M. = I.P.R. * RPM

UNIT: SPEEDS AND FEEDS - COUNTERBORING

$$RPM = \frac{SFPM * 4}{DIA.}$$

SUGGESTED SPEEDS

MATERIAL	SPEED (SFPM)
ALLOY STEEL	40 - 80
BRASS	150 - 300
CAST IRON SOFT	120 - 140
MILD STEEL	75 - 85

SUGGESTED FEEDS

COUNTERBORE DIAMETER	FEED (IPR)
1/4 - 3/8	.003 - .005
7/16 - 5/8	.004 - .006
11/16 - 7/8	.005 - .007
15/16 - 1 3/16	.006 - .008
1 1/4 - 1 1/2	.007 - .009
1 9/16 - 2.0	.008 - .010

UNIT: SPEEDS AND FEEDS - TAPPING

SUGGESTED TAPPING SPEEDS IN R.P.M.

TAP SIZE	CAST IRON	ALUMINUM	MILD STEEL	ST. STEEL	BRASS
10-24	1300	1400	1100	500	1500
1/4-20	1000	1200	750	400	1200
5/16-18	850	1100	650	300	1200
7/16-14	600	800	450	200	950
1/2-13	500	650	400	200	850
5/8-11	375	500	300	190	700
3/4-10	325	400	250	125	575
1 - 8	250	300	175	75	425

SUGGESTED TAPPING FEED RATES

FEED RATE = PITCH OF THREAD * PRM * .95

PITCH = 1 / NUMBER OF THREADS PER INCH

.95 IS 95 % OF VALUE TO ALLOW SOME TAP " FLOATING " DURING CUT.

EX. TAP CAST IRON USING A 1/2 - 13 TAP

P = 1 / N = 1 / 13 = .0769

FEED RATE = .0769 * 500 * .95 = 36.52 I.P.M.

THUS IN YOUR PROGRAM F36.52

UNIT: SPEEDS AND FEEDS - BORING

SUGGESTED BORING SPEEDS IN S.F.P.M.

$$RPM = \frac{S.F.P.M. * 4}{DIA.}$$

RECOMMENDED BORING SURFACE FEET PER MINUTE

	CAST IRON	BRASS	ALUM.	MILD STEEL	TOOL STEEL	STAIN STEEL
H.S.S	70	125	150	100	60	40
CARBIDE	225	300	500	200	100	125

FEEDS FOR ROUGHING

FEEDS FOR ROUGH BORING .007 - .015 INCH PER REV (I.P.R.)

FEEDS FOR FINISH BORING .001 - .005 INCH PER REV. (I.P.R.)

Speeds and Feeds

UNIT : SPEEDS AND FEEDS - END MILLING

MATERIAL	FEED / TOOTH .25 D	FEED / TOOTH .50 D	FEED / TOOTH 1.0 D	S.F.P.M.
ALUMINUM	.003	.006	.009	150
MAGNESIUM	.004	.006	.010	150
BRONZE	.003	.005	.007	250
CAST IRON	.003	.005	.008	60
STEEL LC	.001	.002	.004	80
STEEL ALY	.0005	.001	.003	50
STEEL SS	.001	.002	.004	55
INCONEL	.0002	.001	.003	30
TITANIUM	.001	.002	.004	25

$$\text{RPM} = \frac{\text{S.F.P.M.} * 4}{\text{DIA.}}$$

$$\text{FEED RATE} = R * T * \text{RPM}$$

R = FEED / TOOTH

T = # OF TEETH ON END MILL

RPM = RPM CALCULATED THROUGH FORMULA

Speeds and Feeds

UNIT: FACE MILLING

MATERIAL	DEPTH OF CUT	HSS TOOL		CARBIDE INSERT	
		SPEED	FEED	SPEED	FEED
ALUMINUM	.250	800	.022	MAX	.020
	.025	1200	.010	MAX.	.010
MAGNESIUM	.250	900	.022	MAX	.020
	.025	1500	.010	MAX.	.010
MILD STEEL	.150	175	.012	625	.014
	.025	230	.008	800	.008
SS STEEL TYPE 303	.150	130	.010	415	.014
	.025	160	.008	550	.008

SPEED= SFPM

FEED = IN PER TOOTH

$$\text{RPM} = \frac{\text{SFPM} * 4}{\text{DIA}}$$

FEED RATE = R * T * RPM

R= FEED IN INCH PER TOOTH

T= NUMBER OF TEETH

Speeds and Feeds

Speed and Feed Calculation Using Slide Calculator

PROBLEM # 1

OPERATION - END MILLING MATERIAL - CHILLED CAST IRON

TOOL SIZE- .375 DIA. (HSS) 2 FLUTE

FEED / TOOTH = ____________

RPM = ________________ FEED RATE = ______________________

PROBLEM # 2

OPERATION - FACE MILLING MATERIAL - INCONEL X

TOOL SIZE- 3.5 DIA. (CARBIDE) 4 TEETH

FEED / TOOTH = _______________

RPM = ________________ FEED RATE = ________________________

PROBLEM # 3

OPERATION - FACE MILLING MATERIAL - COPPER

TOOL SIZE- 2.5 DIA. (HSS) 6 TEETH

FEED / TOOTH = _____________

RPM = ________________ FEED RATE = ________________________

PROBLEM # 4

OPERATION - END MILLING MATERIAL - MAGNESIUM ALLOY

TOOL SIZE - .75 DIA. (CARBIDE TOOL) 3 FLUTE

FEED / TOOTH = _________________

RPM = ________________ FEED RATE = ________________________

Speeds and Feeds Questions

Calculate the correct RPM and feed rate for each machining situation below.

1. Operation - center drilling
   Material - cast iron
   #3 C-Drill (H.S.S.)
   RPM]________________
   Feed Rate] ___________

For this problem, use RPM vs. cutting speed conversion chart.

2. Cutter diameter - 1.25
   Cutting speed - 450

   RPM]________________

Speeds and Feeds Questions

Use conversion chart for feed.

3. Feed per tooth - .002
Number of teeth - 8
RPM - 125
Feed Per Minute] ______

4. Operation - spot drilling
Material - cast iron
1/2 diameter (H.S.S.) spot drill

RPM]________________
Feed Rate] ___________

5. Operation - drilling
Material - cast iron
1/4 diameter drill (regular)

RPM]________________
Feed Rate] ___________

Speeds and Feeds Questions

4. Operation - reaming
Material - stainless steel
.500 dia reamer (H.S.S.)
RPM]________________
Feed Rate] ___________

5. Operation - countersinking
Material - cast iron
.75 diameter C-sink
RPM]________________
Feed Rate] ___________

5. Operation - tapping (non-rigid)
Material - cast iron
5/8 - 11 tap
RPM]________________
Feed Rate] ___________

Speeds and Feeds Questions

9. Operation - tapping (rigid)
Material - brass
1-8 tap
RPM]________________
Feed Rate] ____________

10. Operation - boring
Material - cast iron
Carbide tool
Fine boring - .950 dia
RPM]________________
Feed Rate] ____________

Speeds and Feeds Questions

11. Operation - end milling
Material - titanium
.500 dia H.S.S. end mill
RPM]________________
Feed Rate] ____________

12. Operation - face milling
Material - mild steel
5.0 dia carbide
Face mill with 8 inserts
RPM]________________
Feed Rate] ____________

Your Notes:

Section Twelve

Your Notes:

Training Daily Guide

Session Twelve

During this four hour segment, participants will:

1. Understand the features found on a Yasnac control for CNC lathes
2. Understand the features found on a machine control station for CNC lathes
3. Understand axes movement found on a 2 axis CNC lathe
4. Understand common "G" and "M" codes used in programming a CNC lathe
5. Understand special codes of G96 and G99; and how they relate to spindle speed
6. Understand special codes of G96 and G99; and how they relate to feed rate
7. Understand the reasons behind selecting right hand or left hand cut turning tools for CNC lathe operation
8. Understand tool call and offset commands

more on next page

Training Daily Guide

9. Apply the correct programming does for a finishing pass on a turned part

10. Understand how to apply a sub-program within a main program

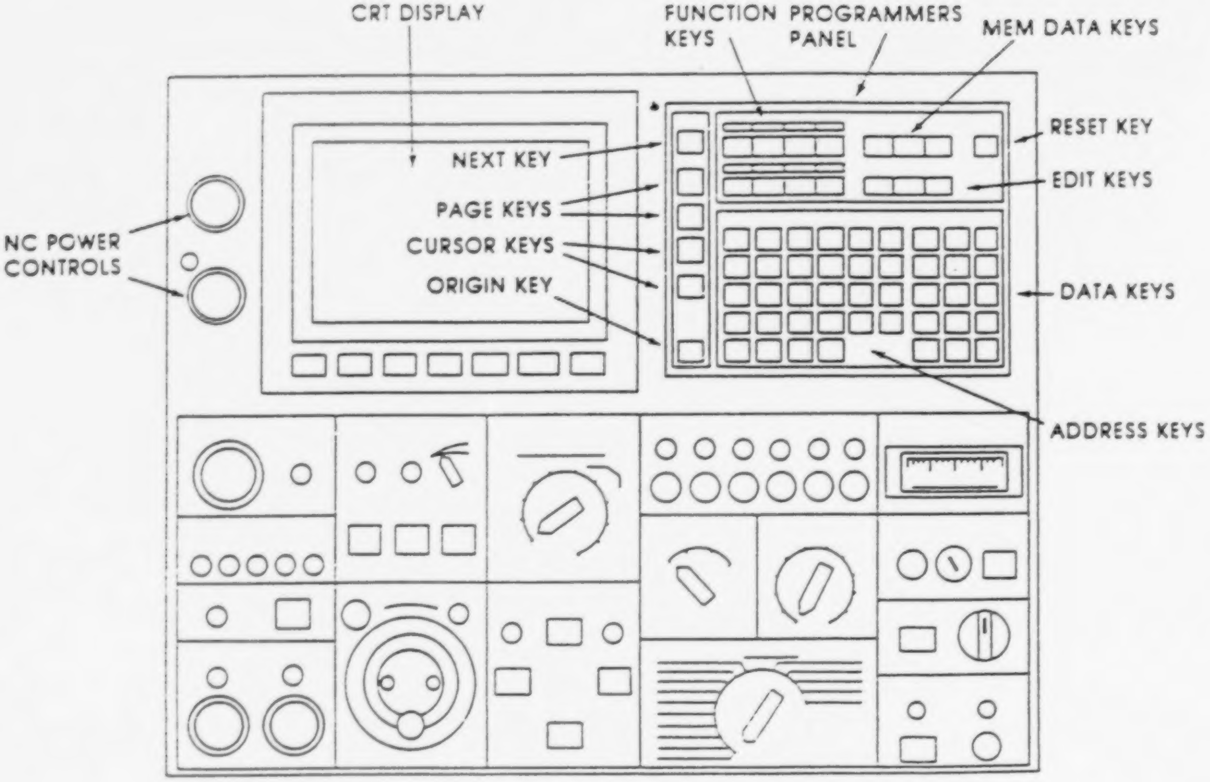
11. Understand basic lathe formulas of conversion factors and production time

12. Understand specific fixed cycles used in CNC lathes

13. Apply knowledge of CNC training center by answering question on worksheets

Nakamora CNC Lathe

Yasnac CNC Control

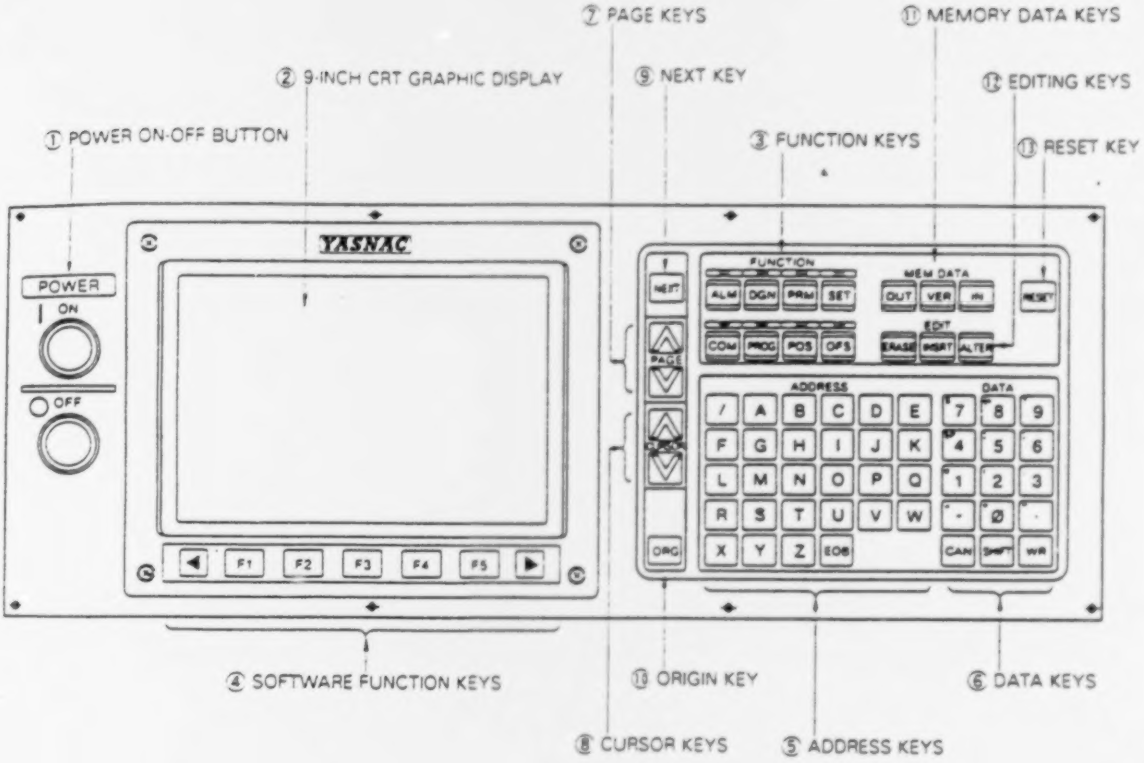

CRT DISPLAY
FUNCTION KEYS
PROGRAMMERS PANEL
MEM DATA KEYS
RESET KEY
NEXT KEY
EDIT KEYS
PAGE KEYS
NC POWER CONTROLS
CURSOR KEYS
DATA KEYS
ORIGIN KEY
ADDRESS KEYS

Yasnac Controller

240

Nakamora CNC Lathe

Yasnac LX3 Control


⑦ PAGE KEYS
⑪ MEMORY DATA KEYS
② 9-INCH CRT GRAPHIC DISPLAY
⑨ NEXT KEY
⑫ EDITING KEYS
① POWER ON-OFF BUTTON
③ FUNCTION KEYS
⑬ RESET KEY
YASNAC
POWER
ON
OFF
F1
F2
F3
F4
F5
NEXT
PAGE
CURSOR
ORG
FUNCTION
ALM
DGN
PRM
SET
MEM DATA
OUT
VER
IN
RESET
COM
PROG
POS
OFS
EDIT
ERASE
INSRT
ALTER
ADDRESS
DATA
/ A B C D E
F G H I J K
L M N O P Q
R S T U V W
X Y Z EOB
7 8 9
4 5 6
1 2 3
- 0 .
CAN
SHIFT
WR
④ SOFTWARE FUNCTION KEYS
⑩ ORIGIN KEY
⑥ DATA KEYS
⑧ CURSOR KEYS
⑤ ADDRESS KEYS

YASNAC LX3 Operator Panel

Nakamora CNC Lathe

Panel Functions.

1. Power On
   Push once to_________________________.
   Push again to_________________________.

   Power Off
   Push once to de-energize both the
   ______________ and the __________ power
   system of the NC unit.

2. CRT graphic display

3. Function keys

(1) ALM	Alarm Key	Displays _______Codes.
(2)DGN	Diagnostic Key	Displays I/O Signal Info.
(3) PRM	Parameter Key	Displays parameters
(4) SET	Setting Key	For setting and writing data.
(5) COM	Command Key	For displaying automatic operation command values.
(6) PROG	Program Key	For part program display and writing
(7) POS	Position Key	For displaying various current values.
(8) OFS	Offset Key	For displaying and writing tool offset values.

Nakamora CNC Lathe

4. Software Function Keys
   Depressing Keys F1 through F5 calls up the corresponding functions.

5. Address Keys
   The keys for keying the address characters when wiring in various data.

   Special Characters

   / Key: For _________________ skip.

   EOB Key: For commanding end of one block. On the CRT, "j" is displayed instead of EOB.

3. Data Keys
   The 15 data keys are used for writing:
   - (1) MDi command values
   - (2) Tool offset values
   - (3) Setting values
   - (4) Parameters
   - (5) Other numerical values

   CAN (Cancel) Key: for deleting working values or address data

   WR (Write) Key: for storing address and data key in values

Shift Key: for inputting special codes found on upper left corner of keys. Depressing the respective key.

Nakamora CNC Lathe

7. Page Keys

∨ Key: For displaying the next page
∧ Key: For displaying the preceding page

8. Cursor Keys

∨ Key: Moves cursor backward
∧ Key: Moves cursor forward

9. Next Key

Used for special functions

1. Selects search type in edit mode
2. Selects between line cursor and word cursor in edit mode

10. Origin Key

For setting the current tool position as the new origin of the new coordinate system

Nakamora CNC Lathe

11. Memory data keys
    OUT, VER, IN keys are effective only in edit mode.

OUT key: For sending a program out from control to a computer, etc.

IN key: For sending a program into control from a computer.

VER key: (Verify) checks data in memory against data sent in by computer or tape reader

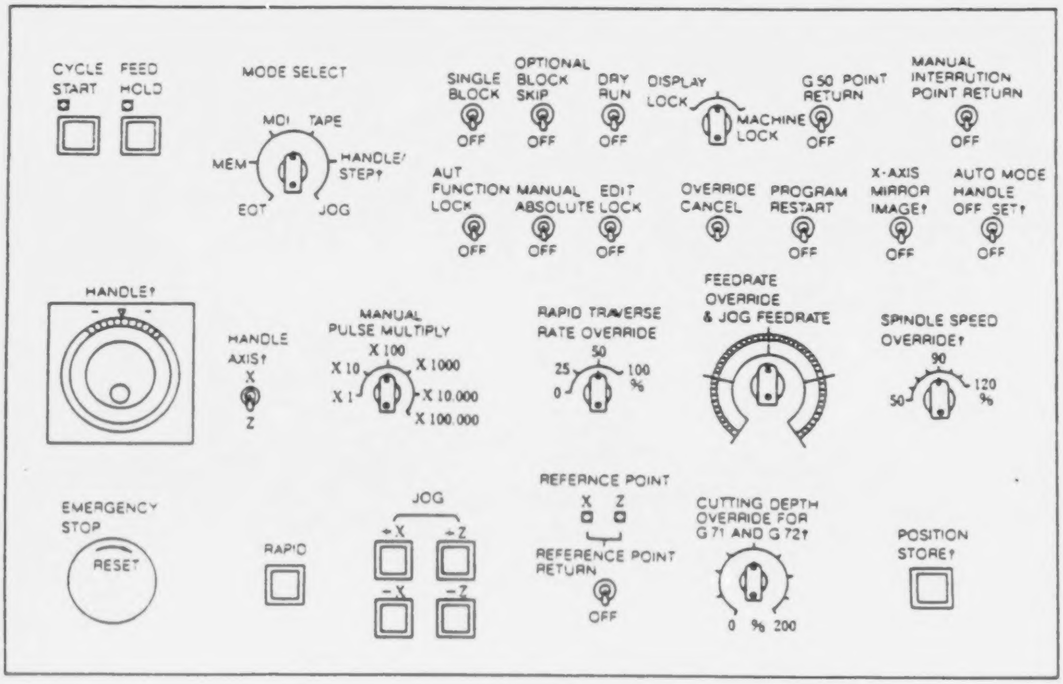
12. Edit keys:
    ERASE, INSRT, ALTER keys are for editing part programs stored in one memory.

ERASE key:	For erasing data in memory
INSRT key:	For inserting data
ALTER key:	For changing data

13. Reset key
    Resets internal condition of the NC
    1. Clears buffer
    2. Clears alarm

Nakamora CNC Lathe

Machine control station


CYCLE START
FEED HOLD
MODE SELECT
MDI
TAPE
MEM
HANDLE/ STEP†
EDT
JOG
SINGLE BLOCK
OFF
OPTIONAL BLOCK SKIP
OFF
DRY RUN
OFF
DISPLAY LOCK
MACHINE LOCK
G 50 POINT RETURN
OFF
MANUAL INTERRUTION POINT RETURN
OFF
AUT FUNCTION LOCK
OFF
MANUAL ABSOLUTE
OFF
EDIT LOCK
OFF
OVERRIDE CANCEL
PROGRAM RESTART
OFF
X-AXIS MIRROR IMAGE†
OFF
AUTO MODE HANDLE OFF SET†
OFF
HANDLE†
HANDLE AXIS†
X
Z
MANUAL PULSE MULTIPLY
X 100
X 10
X 1000
X 1
X 10.000
X 100.000
RAPID TRAVERSE RATE OVERRIDE
50
25
100
0
%
FEEDRATE OVERRIDE & JOG FEEDRATE
SPINDLE SPEED OVERRIDE†
90
50
120
%
EMERGENCY STOP
RESET
RAPID
JOG
+X
+Z
-X
-Z
REFERANCE POINT
X
Z
REFERENCE POINT RETURN
OFF
CUTTING DEPTH OVERRIDE FOR G 71 AND G 72†
0
%
200
POSITION STORE†

BEST COPY AVAILABLE

Nakamora CNC Lathe

Cycle Start - Depress to start system in the automatic mode (tape, MDi, and mem.)

Feed Hold - Temporary stop of automatic operation

1. Not active during thread cutting
2. Depress cycle start to restart operation

Emergency stop push button - stops power

1. Machine stopped immediately by dynamic brake
2. Will display alarm code 330
3. To restart operation
   a] turn E-stop knob clockwise
   b] depress reset
   c] press Power On pushbutton
4. Use for turning off system

Nakamora CNC Lathe

Handle Dial (Manual pulse generator)

1. Set mode to handle
2. Select axis
3. Set mode amount per graduation
4. Rotate dial to move select axis
   CW = + direction
   CCW = - direction

Handle axis select

1. Selects X or Z axis

Manual pulse multiply switch

1. Selects value for handle mode
2. X1 to X100
   X1 = .0001 inch/step
   X100 = 10 inches/step

Jog and rapid pushbuttons

1. Used to feed tool manually
2. +X, -X, +Z, -Z
3. Jog speed set by jog feed rate switch
4. Each time the pushbutton is depressed in the STEP mode, the tool is moved by the value per step by Manual Pulse Multiply select switch.

Nakamora CNC Lathe

Jog feedrate switch and feedrate override switch.

1. In auto mode, feedrate adjustable from 0 - 200% of programmed feedrate.
2. Jog feedrate - used to select feedrate in the jog mode.

Rapid traverse rate override switch

1. Settable to F0, 25, 50 and 100%

Feedrate override cancel switch

1. Turning this switch on, feedrate is fixed at 100%

Nakamora CNC Lathe

Spindle speed override switch

1. Spindle speed adjustable from 50 - 120% of current spindle speed.

Manual reference point return switch

1. Switch used to bring the tool back to reference point manually.

Reference pint lamps

1. Lamps indicate that the tool is positioned on the reference pint
2. G28 auto return to reference point

Single block switch

1. Block-by-block can be obtained when this switch is turned on

Optional block skip switch

1. When On all commands in a block following / are neglected
2. When Off, all blocks including those preceded by / are executed.

Nakamora CNC Lathe

Dry run switch

1. When On, tool moves at speed selected by the JOG FEEDRATE switch

Display lock/machine lock switch

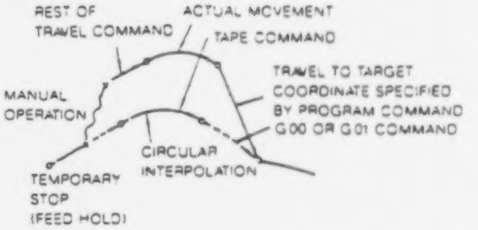
1. When OFF, normal position display
2. When Display Lock excludes tool movement from display
3. When Machine Lock, axis movement including zero return is inhibited

M-Function Lock Switch

1. When ON, it ignores M, S, and T commands
2. To check program, it is used in combination with Machine Lock function

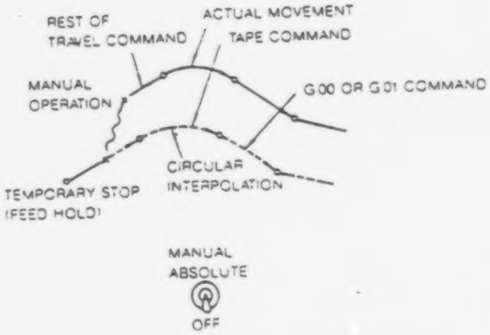
Manual Absolute Switch

1. When On, the tool moves as per diagram below.


REST OF TRAVEL COMMAND
ACTUAL MOVEMENT
TAPE COMMAND
TRAVEL TO TARGET COORDINATE SPECIFIED BY PROGRAM COMMAND
G00 OR G01 COMMAND
MANUAL OPERATION
CIRCULAR INTERPOLATION
TEMPORARY STOP (FEED HOLD)

Tool Movement with MANUAL ABSOLUTE Switch On

2. When Off, the tool moves as per diagram below.


REST OF TRAVEL COMMAND
ACTUAL MOVEMENT
TAPE COMMAND
G00 OR G01 COMMAND
MANUAL OPERATION
CIRCULAR INTERPOLATION
TEMPORARY STOP (FEED HOLD)
MANUAL ABSOLUTE
OFF

Tool Movement with MANUAL ABSOLUTE Switch Off

Nakamora CNC Lathe

Edit Lock Switch

1. When On, the following operations will not work.
    - a] Erase, Insert, and Alter keys
    - b] Storing of NC tape

G50 point return switch

1. Returns tool to the coordinate system setup point manually (where G50 has been programmed)

Manual interruption point return switch

1. When On, returns the tool to where the NC was switched over from the auto mode to the manual mode.

Program Restart Switch

1. Used for restarting the part program from any desired sequence number

Nakamora CNC Lathe

X Axis mirror image

1. When On, all X axis coordinate are reversed.

Auto mode handle offset switch

1. Enables tool motion through the use of MPG knob.

Cutting depth override switch

G71 is a rough turning cycle
G72 is a rough facing cycle

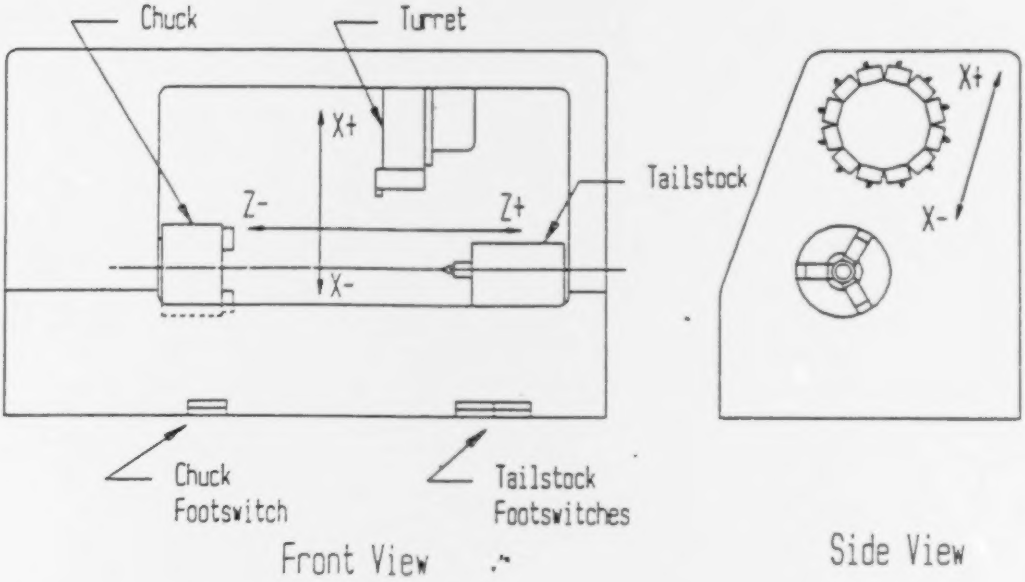
1. Can override depths of cut from 10 - 200%

Positing store pushbutton

1. Used for inputting measured workpiece values direction into control.

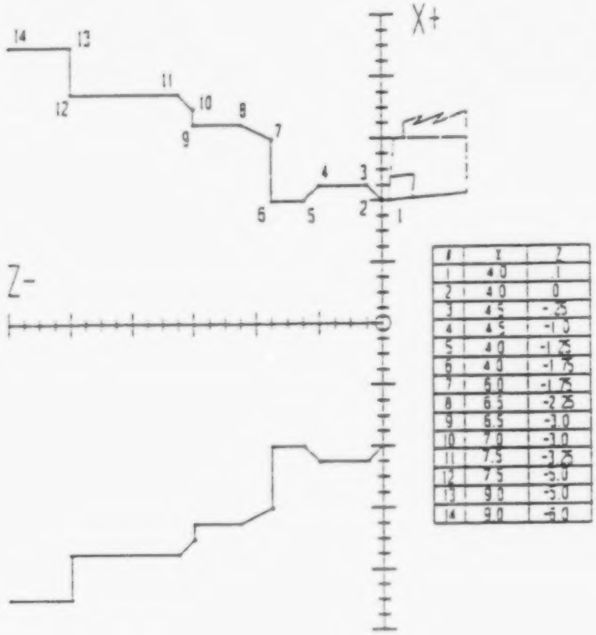
Nakamora CNC Lathe

Universal slant bed turning center


Chuck
Turret
X+
Z-
Z+
X-
Tailstock
X+
X-
Chuck Footswitch
Tailstock Footswitches
Front View
Side View

All major applications for turning can be accomplished with this form of turning center including; shaft work (with tailstock), chucking work, and bar work (if a bar feeder is used).

Nakamora CNC Lathe


X+
Z-
14
13
12
11
10
9
8
7
6
5
4
3
2
1

#	X	Z
1	4.0	.1
2	4.0	0
3	4.5	-.25
4	4.5	-1.0
5	4.0	-1.25
6	4.0	-1.75
7	6.0	-1.75
8	6.5	-2.25
9	6.5	-3.0
10	7.0	-3.0
11	7.5	-3.25
12	7.5	-5.0
13	9.0	-5.0
14	9.0	-6.0

X axis is a diameter controlling axis.

Z axis is a length (depth) controlling axis.

List of G Codes

B: Basic
O: Optional

G Code	Special G Code I	Special G Code II	Group	Function	Section
◤ G 00	◤ G 00	◤ G 00	01	Positioning (rapid traverse feed)	B
◤ G 01	◤ G 01	◤ G 01		Linear interpolation, angle programming for linear interpolation	B, O
G 02	G 02	G 02		Circular interpolation CW, (radius R designation)	B, O
G 03	G 03	G 03		Circular interpolation CCW, (radius R designation)	B, O
G 04	G 04	G 04	*	Dwell	B
G 06	G 06	G 06		ERROR DETECT OFF positioning	B
G 10	G 10	G 10		Tool offset value setup	O
G 11	G 11	G 11	01	Beveling Cornering	O
G 12	G 12	G 12		Rounding	O
G 20	G 20	G 70	05	Inch input specification	O
G 21	G 21	G 71		Metric input specification	O
G 22	G 22	G 22	01	Radius programming for circular interpolation CW	O
G 23	G 23	G 23		Radius programming for circular interpolation CCW	O
G 27	G 27	G 27	*	Reference point return check	B
G 28	G 28	G 28		Automatic return to reference point	B
G 29	G 29	G 29		Return from reference point	B
G 30	G 30	G 30		Returnn to 2nd reference point	O
G 31	G 31	G 31		Skip function	O
G 32	G 33	G 33	01	Threadcutting, continuous threadcutting, multi-start threadcutting	B, O
G 34	G 34	G 34		Variable lead threadcutting	O
G 35	G 35	G 35	*	Tool set error compensation	O
G 36	G 36	G 36	07	Stored stroke limit 2nd area ON	O
G 37	G 37	G 37		Stored stroke limit 2nd area OFF	O
G 38	G 38	G 38	08	Stored stroke limit 3rd area ON	O
G 39	G 39	G 39		Stored stroke limit 3rd area OFF	O
◤ G 40	◤ G 40	◤ G 40	06	Tool radius compensation cancel	O
G 41	G 41	G 41		Tool radius compensation No. 1	O
G 42	G 42	G 42		Tool radius compensation No. 2	O
G 43	G 43	G 43		Tool radius compensation No. 3	O
G 44	G 44	G 44		Tool radius compensation No. 4	O

Note:

1. G codes in groups from 01 through 11 are modal. When the control is energized with the power switch or reset, the G codes marked with ◤ are automatically selected.

   For G 00/G 01, G 98/G 99, and G 90/G 91, either one is selected as initial state by setting parameters.
2. G codes of * group are non-modal. They should not be commanded together with the other G codes in one block.
3. The modal G codes can be commanded mixedly in a block.
4. G codes in section B are basic.
5. Standard G codes can be converted to special G codes I by parameters. (basic feature)
6. Special G code II can be selected as optional function. When selected, the standard G codes and special G code II cannot be used.
7. The initial states of G codes of 05, 07, 08 group when the control is powered correspond to their respective setting data.

List of G Codes

G Code	Special G Code I	Special G Code II	Group	Function		Section
G 50	G 92	G 92	*	Coordinate system setup		B
				Maximum spindle revolution setup, work coordinate system setup		○
G 51	G 51	G 51	*	Return of current display value to origin		○
G 65	G 65	G 65		User macro simple call		○
G 66	G 66	G 66	09	User macro modal call		○
◤ G 67	◤ G 67	◤ G 67		User macro modal call cancel		○
G 68	G 68	G 68	10	Mirror image by programming ON		○
◤ G 69	◤ G 69	◤ G 69		Mirror image by programming OFF		○
G 70	G 70	G 72	*	Finishing cycle	Multiple repetitive cycles	○
G 71	G 71	G 73		Stock removal in turning		○
G 72	G 72	G 74		Stock removal in facing		○
G 73	G 73	G 75		Pattern repeating		○
G 74	G 74	G 76		Peck drilling in Z-axis		○
G 75	G 75	G 77		Grooving in X-axis		○
G 76	G 76	G 78		Automatic threadcutting cycle		○
G 90	G 77	G 20	01	Turning cycle A		B
G 92	G 78	G 21		Threading cycle		B
G 94	G 79	G 24		Facing cycle B		B
G 96	G 96	G 96	02	Constant surface speed control		○
◤ G 97	◤ G 97	◤ G 97		Constant surface speed control cancel		○
◤ G 98	◤ G 94	◤ G 94	04	Feed per minute (mm/min)		B
◤ G 99	◤ G 95	◤ G 95		Feed per revolution (mm/rev)		B
	◤ G 90	◤ G 90	03	Absolute command		B
	◤ G 91	◤ G 91		Incremental command		B
G 122	G 122	G 122	11	Tool registration start	Tool life control	○
◤ G 123	◤ G 123	◤ G 123		Tool registration end		○
G 111	G 111	G 111	*	Taper multiple beveling/rounding		○
G 112	G 112	G 112		Arc multiple beveling/rounding		○
G 150	G 150	G 150	13	Cancel groove width compensation		○
G 151	G 151	G 151		Groove width compensation		○

Note:

1. G codes in groups from 01 through 11 are modal. When the control is energized with the power switch or reset, the G codes marked with ◤ are automatically selected.

   For G 00/G 01, G 98/G 99, and G 90/G 91, either one is selected as initial state by setting parameters.

2. G codes of * group are non-modal. They should not be commanded together with the other G codes in one block.

3. The modal G codes can be commanded mixedly in a block.

4. G codes in section B are basic.

5. Standard G codes can be converted to special G codes I by parameters. (basic feature)

6. Special G code II can be selected as optional function. When selected, the standard G codes and special G code II cannot be used.

7. The initial states of G codes of 05, 07, 08 group when the control is powered correspond to their respective setting data.

Typical M Codes

M code	Description
M00	Program stop
M01	Optional stop
M02	End of program (does not rewind program)
M03	Spindle on in a clockwise direction
M04	Spindle on in a counter clockwise direction
M05	Spindle stop
M08	Flood coolant on
M09	Coolant off
M10	Step part counter
M13	Bar feeder on
M14	Chuck jaws close
M15	Chuck jaws open
M16	Tailstock quill forward
M17	Tailstock quill back
M28	Tailstock body forward
M29	Tailstock body back
M41	Low spindle range
M42	High spindle range
M30	End of program (rewinds memory)
M98	Subprogram call
M99	End of subprogram

Nakamora CNC Lathe

Spindle Speed Choices

I. Constant surface speed [G96]
   A. Use for operations:
      1] Contouring operations (turning)
      2] Boring operations
      3] Grooving tools
   B. Feedrate in I.P.R. remains constant
   C. Improved tool life

example: N005 G96 X300
(Selects speed of 300 S.F.P.M.)

II. RPM [G97]
   A. Use for operations
      1] Machining on spindle centerline
         a. drilling
         b. reaming
         c. tapping
      2] Threading
         a. RPM and Feedrate must be synchronized

example: N005 G97 S300
(Selects speed of 300 RPM)

Nakamora CNC Lathe

Feedrate Choices

I. Inches per revolution (i.p.r.)
  A. Maintains even witness marks on workpiece
  B. Use for almost all turning center operations

example: *N010 G99 F.015*
(Selects .015 inches per revolution)

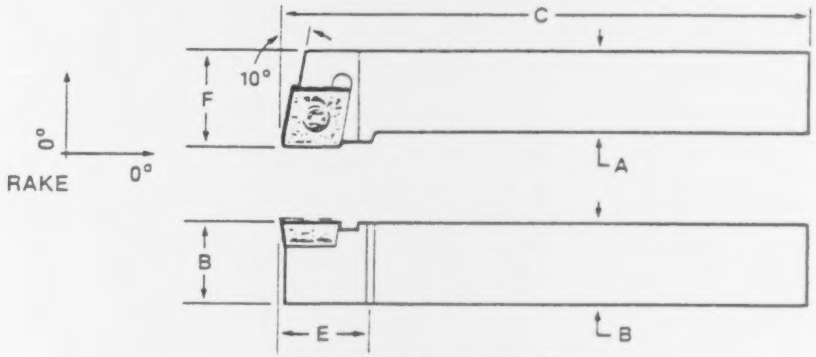
II. Inches per minute
  A. Some preset for knurling
  B. Use for stopping spindle
  (i.e. bar feed move)

example: *N010 G98 F 5- 0*
(Selects 5 inches per minute)

Turning Tools

Spindle Direction

M03	CW	For R.H. cut tools
M04	CCW	For L.H. cut tools



RIGHT HAND SHOWN

Example of right hand tool holder.

T Codes

Example: T0505

T05 = Turret station #5
05 = Offset number 5 (wear offset)

Note: Tool coordinate offset is derived by adding 50 to the value of index position. Thus, for position of the offset coordinate is 51.

Geometry Offsets = Used to designate program zero location

Wear Offsets = Workpiece sizing and tool nose radius compensation

Nakamora CNC Lathe

Most CNC Controls have at least 16 offsets. Some have 32, 99 and more.

The offset table may look like this.

#	X	Z	R	T
______	______	______	______	______

Where:

#____

X____

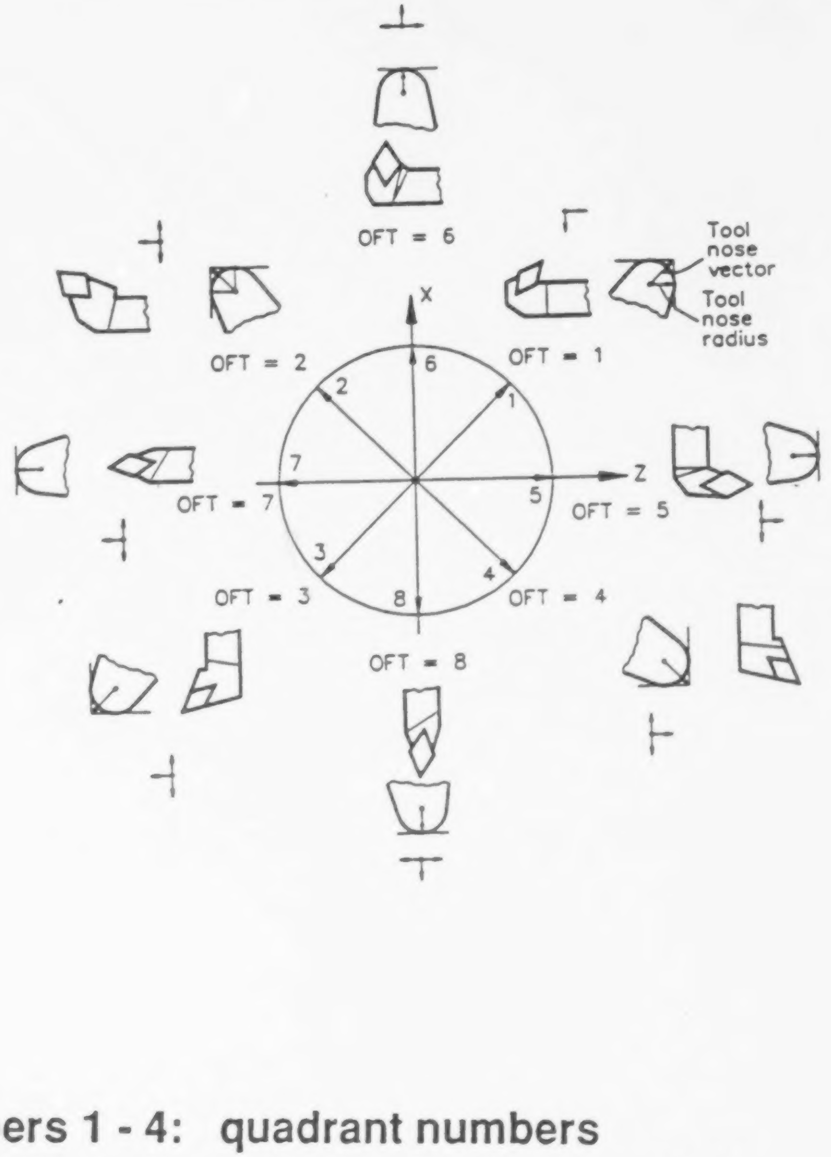
Z____

R____

T____

Nakamora CNC Lathe

Tool offset (T) in tool offset register direction of imaginary tool nose

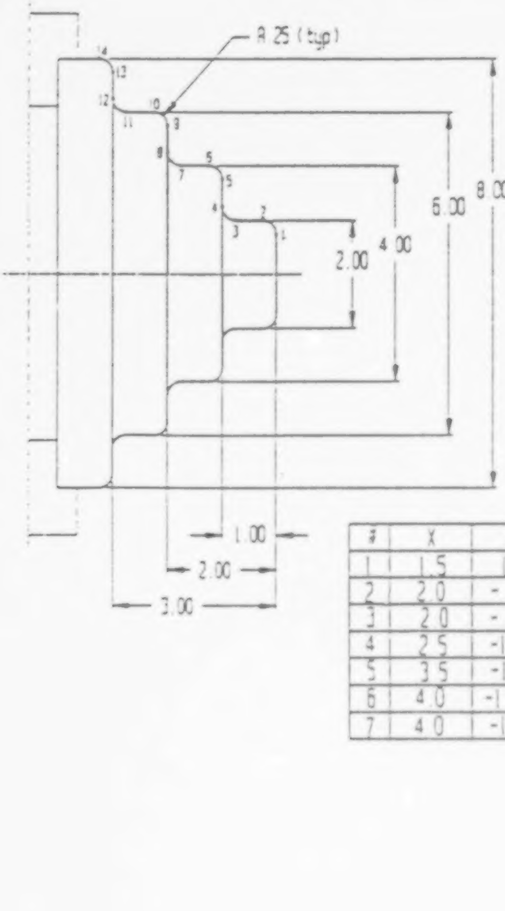

OFT = 6
Tool nose vector
Tool nose radius
OFT = 2
OFT = 1
X
6
2
1
7
Z
5
OFT = 7
OFT = 5
3
4
8
OFT = 3
OFT = 4
OFT = 8

Numbers 1 - 4: quadrant numbers
Numbers 5 - 8 point to +Z, +X, -Z, and -X

Nakamora CNC Lathe

Programming a turned part

For the part sketch shown below, write a NC program to finish turn this part.


R.25 (typ)
14
13
12
11
10
9
8
7
6
5
4
3
2
1
2.00
4.00
6.00
8.00
1.00
2.00
3.00

#	X	Z
1	1.5	0
2	2.0	-.25
3	2.0	-.75
4	2.5	-1.0
5	3.5	-1.0
6	4.0	-1.25
7	4.0	-1.75
8	4.5	-2.0
9	5.5	-2.0
10	6.0	-2.25
11	6.0	-2.75
12	6.5	-3.0
13	7.5	-3.0
14	8.0	-3.25

Program

00002 (Program Number)

N005____________________

N010____________________

N015____________________

N020____________________

N025____________________

N030____________________

N035____________________

N040____________________

N045____________________

N050____________________

N055____________________

N060____________________

N065____________________

N070____________________

N075____________________

N080____________________

N085____________________

N090____________________

N095____________________

N100____________________

Nakamora CNC Lathe

Sub-program example used for warn up of turning center.

Main program

```
00001
```

(overnight dwell program for spindle warm up)

```
N005 M98 P1000 L6
```

L work specified the number of hours of dwell

```
N010 G97 S1500 M03
```

Turn spindle on at 1500 RPM

```
N020 M30
```

End of program

Sub Program

```
01000
```

Actual dwelling program

```
N1G04 X3600
```

Dwell for 3600 seconds, one hour

```
N10 M99
```

Return to main program

Nakamora CNC Lathe

Basic lathe formulas

Conversion factors

To convert from "inches per revolution" (ipr) to "Inches per minute" (ipm), or vice versa, use the following:

ipr* = RPM = ipm
ipr = ipm/RPM

To convert from "surface feet per minute" (SFM) to "RPM" or vice versa, use the following:

$$\text{SFM} = \frac{3.1416 \text{ X RPM X dia.}}{12}$$

$$\text{RPM} = \frac{12 \text{ X SFM}}{3.1416 \text{ X dia.}}$$

Nakamora CNC Lathe

Determining time required to perform an operation:

A. Find RPM

$$\text{RPM} = \frac{12 \times \text{SFM}}{3.1416 \times \text{dia.}}$$

B. Time in seconds

$$\text{Time (secs)} = \frac{\text{Distance to go} \times 60 \text{ secs.}}{\text{ipr} \times \text{RPM}}$$

Solve the following problem:
How long will it take to turn a part in one pass to a one inch diameter at 350 SFM and .004 ipr over 3 inches?

Answer: RPM ________
Time in seconds________

Special Fixed Cycles

1. Special fixed cycle

G70: Finishing cycle (effective after G71, G72, G73)

G71: Outer diameter rough cutting cycle

G72: Face rough cutting cycle

G73: Closed loop cutting cycle

……… Cutting feed is done while repeating a certain cutting pattern.

G74: Face cutting-off cycle

……… Effective during drilling

G75: Outer diameter cutting-off cycle

G76: Threading cutting cycle

(1) Finishing cycle

After rough cutting by G71, G72 and G73, finishing cycle can be performed by the following command.

Command system

G70 P_____(ns) Q_____(nf)

P: Sequence number at the beginning of cycle (ns)

Q: Sequence number at the end of cycle (nf)

Application example is given together with G71, G72 and G73.

(Note 1) With F and S code, codes commanded between sequence number ns and nf become effective.

(Note 2) Tip nose radius compensation is possible with this cycle.

(Note 3) Sequence number and block designated by P and Q should be placed before the block of G70.

(Note 4) Program sequence number at the beginning of the block of G70.

G17 Rough Cutting Cycle

(2) G71 Outer diameter rough cutting cycle

This cycle is achieved by designating cutting conditions and finishing dimensions.

Command system

```
G71 P____(ns) Q____(nf) U____(Δu) W____(Δw) D____(Δd) F____(f) S____(s)
```

P : Sequence number at the beginning of cycle (ns)

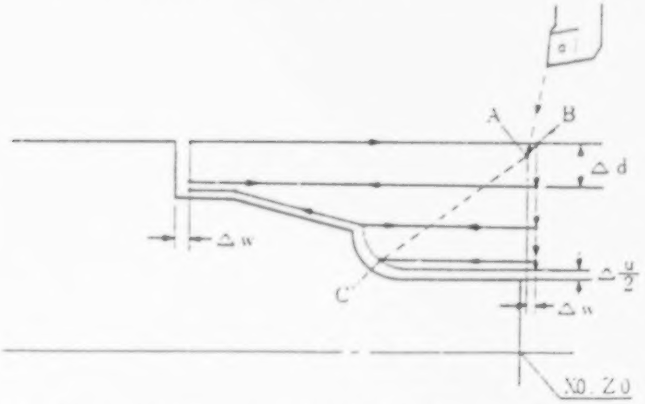
Q : Sequence number at the end of cycle (nf)

U : Distance and direction of cutting tool relief in X axis direction (Δu) ...... Diameter designation ...... Finishing allowance

W : Distance and direction of cutting tool relief in Z axis direction (Δw) ...... Finishing allowance

D : Cutting feed amount/each time (Δd) ...... radius designation ...... Nothing to do with + and -

F·S: F or S function is ignored when it is placed in P or Q block. F or S function which is placed in the block before G71 is effective.



(Note 1) F or S function cannot be changed while G71 cycle is in operation.

(Note 2) Tip nose radius compensation is not effective for this cycle.

(Note 3) Multi-quadrant cutting is not possible with this cycle.

(Note 4) Program sequence number at the beginning of the block of G71.

G73 Closed Loop Cutting Cycle

(4) G73 Closed loop cutting cycle

This cycle is achieved by designating cutting conditions and finishing dimensions.

Command system

G73 P____(ns) Q____(nf) I____(i) K____(k) U____(Δu) W____(Δw) D____(Δd) [illegible]____(t) S____(s)

P: Sequence number at the beginning of cycle (ns)

Q: Sequence number at the end of cycle (nf)

I: Distance and direction of cutting tool relief in X axis direction (i) ----- Radius designation

K: Distance and direction of cutting tool relief in Z axis direction (k)

U: Finishing allowance in X axis direction (Δu) ---- Diameter designation

W: Finishing allowance in Z axis direction (Δw)

D: Number of cutting

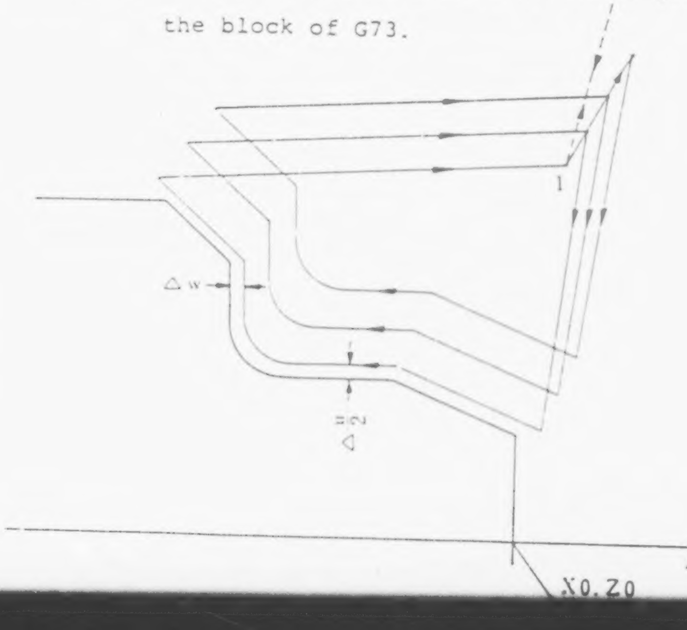
(Note 1) S function cannot be changed while G73 cycle is in operation.

(Note 2) Tool tip "R" compensation is not effective for this cycle.

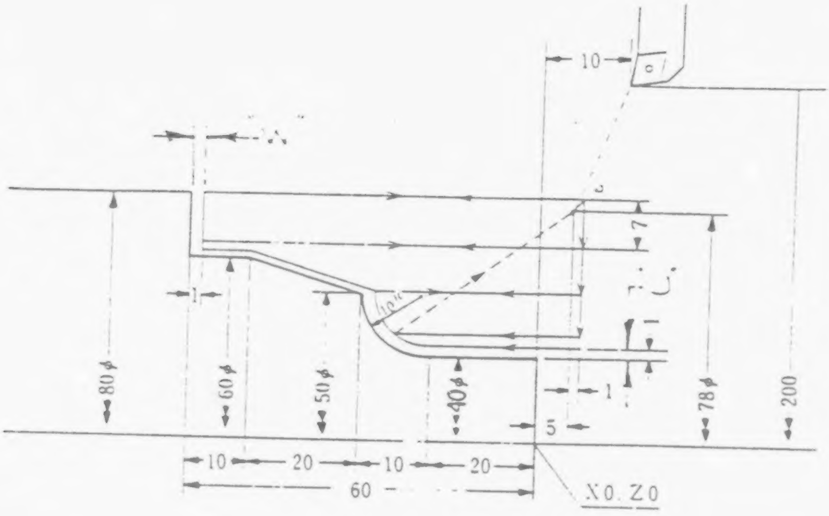
(Note 3) Multi-quadrant cutting is not possible with this cycle.

(Note 4) Four different cutting shapes are produced according to the mark used. Therefore, care should be taken in programming.

(Note 5) Program sequence number at the beginning of the block of G73.



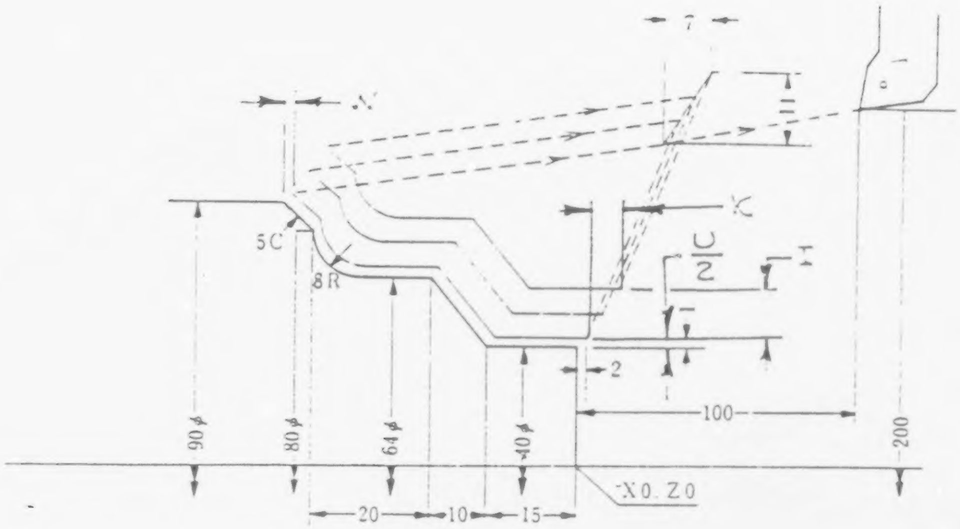
G70 G71 Example


10
"A"
7
1
80φ
60φ
50φ
40φ
1
5
78φ
200
10
20
10
20
60
X0. Z0

```
n001g50x-20000z1000
n002g00t0101  (M41 or M42)
n003g96s400m03
n004g00x-7800z500m08
n005g71p006q011u200w100d700f30
n006g00x-4000
n007g01z-2000f20
n008g03x-5000z-3000i-1000
n009g01x-6000z-5000
n010z-6000
n011x-9000
n012g00x-20000z1000
n013t0100
n014g50x-20000z1000
n015t0202
n016g96s500m03
n017g70p006q011
n018g00x-20000z1000m09
n019t0200
n020m02 (or M30)
```

BEST COPY AVAILABLE

G70 G73 Example



```
n001g50x-20000z10000
n002g00t0101 (M41 or M42)
n003g96s300m03
n004g00x-12000z1000m08
n005g73p006q011i1000k500u200w200d3f30
n006g00x-4000z200
n007g01z-1500f20
n008x-6400z-2500
n009z-3700
n010g03x-8000z-4500i-8000
n011g01x-9000z-5000
n012g00x-20000z10000
n013t0100
n014g50x-20000z10000
n015t0202
n016g96s350m03
n017g70p006q007
n018g00x-20000z10000m09
n019t0200
n020m02 (or M30)
```

Multiple Repetitive Cycles

1. G73 closed loop waiting cycle is also referred to as "contour repetive cycle"

2. G70 can be regarded as a single-pass contouring cycle. It cannot be used alone; it must be used with one of the three stock removal cycles.

3. G73 is best suited for some for GED and casted workpieces that have been made to their rough shapes in which the dimensions are only slightly larger than the finished part size. Use of G71 on this part type would not be efficient because the diameter of the workpieces at various section is not the same.

4. ______________________________

5. ______________________________

Turning Center Worksheet 1

1) Name the two most basic operation panels on a CNC turning center.

____

____

For the following descriptions, name the control panel button which activates the feature.

2) Turns on power to the control panel.

____

3) Turns on power to the machine unit.

____

4) Turns off power to the machine unit.

____

5) Shows display screen position display.

____

6) Shows the active program.

____

7) Shows the offset table.

____

8) Enters data except when editing programs.

____

9) Allows program data to be changed.

____

10) Allows new program data to be entered.

____

11) In the edit mode, this button rewinds the program back to the beginning.

____

12) This series of buttons is on some Fanuc controls below the display screen.

____

Turning Center Worksheet 2

For the following descriptions, name the machine panel button which activates the feature.

1) This multi position switch is the heart of the CNC control.

____

2) This button stops axis motion (only).

____

3) This button activates the program.

____

4) This multi position switch controls the motion rate in the cutting mode.

____

5) This switch controls the motion rate in the rapid mode.

____

6) This switch gives the operator total control of all movements the machine makes.

____

7) This switch makes the machine stop after every command.

____

8) When this switch is on, absolutely no axis motion can occur.

____

9) This switch works in conjunction with an M01.

____

10) This device allows the operator to manually move the axes with a dial.

____

11) This device allows the operator to jog the axes.

____

12) This meter allows the operator to monitor how much stress the spindle is currently under

____

Turning Center Worksheet 3

1) Name the three most basic modes of operation.

____

____

____

2) Which mode switch positions are included in each mode?

____

____

____

3) In which mode does the machine behave the most like an engine lathe?

____

4) In which mode does the operator enter commands through the display screen and keyboard?

____

5) In which mode is the operator actually running workpieces?

____

6) In which mode is the operator editing programs?

____

279

Turning Center Worksheet 4

1) What is the G code used to specify a dwell?

____

2) What happens during a dwell command?

____

____

____

3) What are the three words that can be used to specify the length of time in a dwell command?

____

____

____

4) What word is used to specify an optional block skip command?

____

5) When can the optional block skip command be used?

____

____

____

6) What word is used to specify the radius required when automatic corner rounding and chamfering is used?

____

7) What are the two words used to specify chamfers when automatic corner rounding and chamfering is used. What are their functions?

____

____

Turning Center Worksheet 5

1) What are the four words involved with subprogramming and what are their meanings?

2) What is the most basic purpose for using subprogramming techniques.

3) Name two possible applications for subprograms.

4) What command can be used to end a main program that will not cause the cycle to stop (commonly used for bar feeding programs)?

Your Notes:

Section Thirteen

Your Notes:

Training Daily Guide

Session Thirteen

During this segment, participants will:

1. Review troubleshooting items pertaining to CNC machining.

CNC Troubleshooting

Problems Caused by tooling:

1. Dimensions
2. Finish of workpiece
3. Number of parts

What is tooling:

1. Workholding device
2. Tool holder
3. Clamps
4. Other ______________________________

____

CNC Troubleshooting

Cutting Tool Problems:

What is tooling:

1. Edge (Blue or Burned)
2. Nicked/chipped
3. Rake - material buildup
4. Out of round or bent

Chatter - indicator of trouble.

CNC Troubleshooting

Causes of Chatter

1. Dull tool - most common problem
2. Newly sharpened tool - overly sharp tool
3. Feed rate - increase or decrease
4. Spindle speed - change speed

Example: Feed of F4.0 i.p.m.
Works best at 70% override
4.0 * .70 = 2.8
Thus 2.8 (F2.8) is feed rate to be using.

5. Loose fixture or workpiece

CNC Troubleshooting

Causes of Chatter

6. Lack of coolant
7. Wrong tool used (HSS)
8. Spindle rotation wrong
9. Wrong canned cycle (G81)

Operator Input

CNC Troubleshooting

Operator Input

Your Notes:

Basic CNC Operation

Assessment & Training Guide

Welcome to the Assessment

Welcome to the Dana C.N.C. Operator Assessment Exam.

You are taking this examination to demonstrate your skills in the areas of:

- Shop math
- Blueprint reading and specification sheet reading
- Reasoning skills including:
  - Logical start-up and shut-down procedures
  - Measuring instrument identification
  - Tool identification

This assessment was developed by a team representing Dana C.N.C. operators, supervisors, training staff and management with Anoka-Hennepin Technical College.

Your Goal

Your goal is to correctly answer 75% of more of the questions to receive a passing score.

Area of Study:	***Questions:***	***Your Goal:***
Math	Questions 1 - 30	**22** problems correct
Blueprint reading	Questions 31 - 76	**34** problems correct
Reasoning skills	Questions 77 - 97	**15** problems correct

Passing each section...
After you take the three sections of this exam, you will be asked to demonstrate your hands-on machining skills in a 30 to 45 minute assessment on the shop floor during your regular shift.

An instructor, selected by Anoka-Hennepin Technical College's (AHTC) Custom Training Division, will assess your skills in a consistent and fair manner.

Passing Each Section

If you do not pass the math section:

- You will be asked to take a mathematics refresher course at Dana, presented by AHTC

If you do not pass the blueprint reading section:

- You will be asked to take a blueprint reading course at Dana, presented by AHTC

If you do not pass the reasoning skills section:

- You will be asked to take a C.N.C. theory and hands-on course at Dana, presented by AHTC

If you do not pass the shop floor assessment:

- You will be asked to take a C.N.C. theory and hands-on course at Dana, presented by AHTC

Section 1:
Math
Skills

Math Skills

1. Add

13
33
125
109
______

2. Add

47111
3134
256
9989
______

Math Skills

3. Subtract

```
   156
–   34
_______
```

4. Subtract

```
 1,564
–  198
_______
```

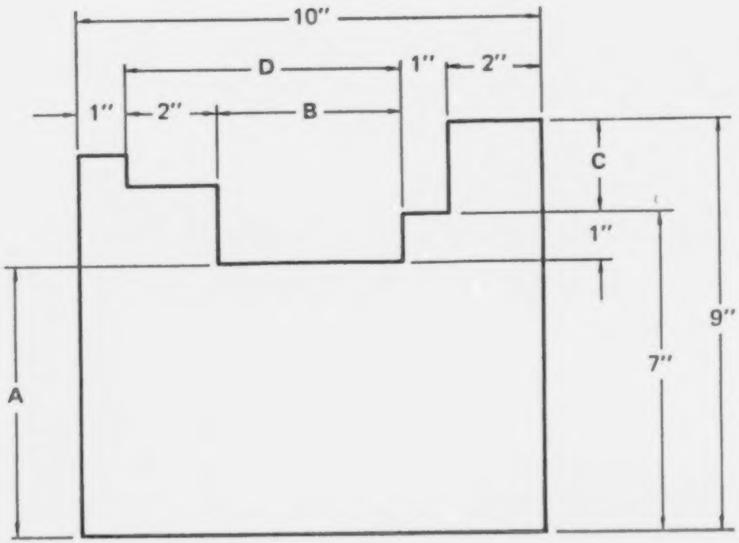
5. Find the lengths of A, B, C, & D

A________

B________

C________

D________



10"
D
1"
2"
1"
2"
B
C
1"
9"
7"
A

Math Skills

6. Multiply

$$\begin{array}{lr} & 93 \\ \times & 12 \\ \hline & \end{array}$$

7. Solve through Multiplication

A machine screw has 8 threads to the inch. How many threads are there in a threaded piece 7 inches long?

____

8. Solve through Division

A machine shop owner spends $2,368 to buy tools. Each tools costs an average of $16. How many tools can be bought?

____

Math Skills

10.

```
   .01
— .015
______
```

11.

```
  - .200
+   .210
________
```

Math Skills

12. Circle your answer

Which is greater?

.015 or -.215

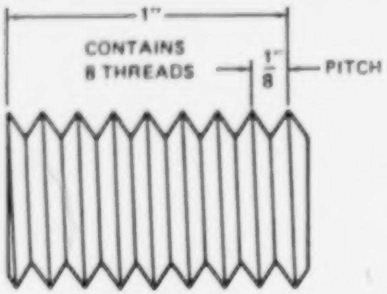
13. Circle your answer

Which is greater?

-.024 or -.026

Math Skills

The pitch of a thread is the distance from a given point on one thread to the corresponding point on the next thread. Pitch is usually expressed as a fraction. Machinists sometimes make the mistake of calling the number of threads in one inch the pitch. For example, a one-inch screw has 8 threads to the inch. The pitch of this thread is 1/8 inch. It is wrong to say that it is an 8 pitch thread.



THREAD PITCH

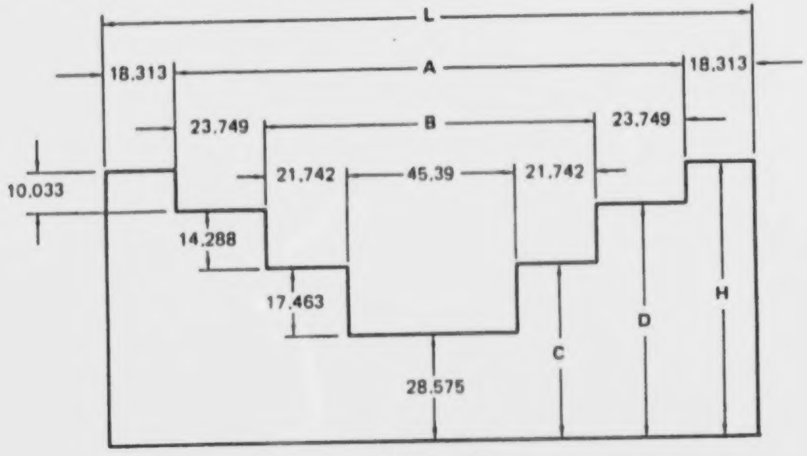
For example only.

14. A screw 4 inches long has 52 threads.
    a. How many threads per inch are there? ______________________
    b. What is the pitch? ______________________________

15. A screw is 3 1/2 inches long and has 56 threads.
    a. Find the number of threads per inch. ____________________
    b. Find the pitch of the screw. __________________________

Math Skills

Note: Use this diagram for problems 16-21.


L
18.313
A
18.313
23.749
B
23.749
10.033
21.742
45.39
21.742
14.288
17.463
28.575
C
D
H

TEMPLATE

16. What is the total length of the template (dimension **L**)?________________

17. What is the total height **H**?______________________________________

18. Find dimension **A**. ____

19. What is dimension **B**? ____

20. Find dimension **C**. ____

21. What is dimension **D**? ____

Math Skills

22. Subtract

$$\begin{array}{lr} & 56.9 \\ - & 3.6 \\ \hline \end{array}$$

23. Subtract

$$\begin{array}{lr} & 9883.456 \\ - & 298.179 \\ \hline \end{array}$$

24. Multiply

$$\begin{array}{lr} & 3.4567 \\ \text{X} & 3.9876 \\ \hline \end{array}$$

25. Divide

$$5.35\,\overline{)\,29.425}$$

Math Skills

26. Express theses subdivisions of an inch in machinists' terms.

Two problems have been done for you.

.000001 Millionth

.00001 ______________________

.0001 ______________________

.001 ______________________

.01 Hundredth

27. Convert the fraction 7/8 to a decimal number __________

28. Convert .75 to a fraction

____

Math Skills

29. Circle the correct answer.

You have a 1 5/16 - 12 tap and you need to go one turn deeper, what numerical value would you assign?

a) -.083
b) .083
c) .016

30. Circle the correct answer.

You have a 1 5/16 - 12 tap and you're two turns shallow, what numerical value would you assign?

a) .317
b) .166
c) -.166

Section 2: Print Reading Skills

Print Reading Skills

Use the folded print found in your assessment package.

31. Identify the 4 views on the print: Write: **top, bottom, end, back.** The **front** view is the second one down on the left side of the print.

32. What is the name of the casting?

____

33. What reference would you use to determine what the casting is made out of?

____

34. What is the decimal tolerance of this print?

____

____

35. What is the latest revision?

____

36. What is the scale of the print?

____

37. Draw a finish symbol in the space provided

38. When a finish is not specified on a view, what finish is required?

____

Print Reading Skills

– A –

Throughout this assessment, symbols, like the one above, will be shown as: **–A–**

39. What features determine **–A–**?
Hint: draw the symbols below

40. What is the starting depth from **–A–**?
____

41. How would you establish **–B–**?
Hint: draw the symbols below

42. What is the overall thickness of the part?
____

43. What is the inlet/port SAE #16 location?

From **–E–** ____________________
From **–H–** ____________________

44. What number would you use to determine the correct Gresen spec sheet to reference this inlet/port?
____

Print Reading Skills

Use the correct Gresen spec sheet, provided in your packet, to help you with the following questions.

45. What is the size of thread SAE #16?

____

46. What does the 12 in your answer above represent?

____

____

47. Thread SAE #16 is .750 deep. How many turns are required?

____

48. What is the minor thread diameter of SAE #16? *Use your spec sheet*

____

49. What is the **U** diameter of SAE #16?

____

50. What is the tolerance zone of the **U** diameter?

____

Print Reading Skills

51. ***Look at the relief Eng Spec 2.003.049 on the print*** What is the location of the center of the relief.

From **–E–** ____________________

From **–A–** ____________________

52. On the back view, how many counterbores [C'BORE] are shown?

____

53. What is the diameter of the counterbores?

(Show minimum and maximum)

____

54. What is the minimum and maximum depth of the counterbores?

____

55. How many 3/8-16 UNC - 2B (bolt holes) are shown on the front view?

____

Print Reading Skills

56. What is the minimum and maximum *drill* depth of 3/8-16 UNC - 2B (bolt holes) on the front view?

____

57. What is the minimum and maximum *thread* depth of 3/8-16 UNC - 2B (bolt holes) on the front view?

____

58. How many 7/16-14 UNC - 2B (bolt holes) are shown on the back view?

____

59. What is the minimum and maximum *drill* depth of 7/16-14 UNC - 2B (bolt holes) on the back view?

____

60. What is the minimum and maximum *thread* depth of 7/16-14 UNC - 2B (bolt holes) on the back view?

____

61. How many Trepan Grooves are located on the:

Front view____________________
Back view____________________

62. Are all the Trepans the same on the front and back views?

____

Print Reading Skills

63, 64, 65, 66.

List the spec numbers and the quantity of Trepan Grooves located on the print.

Front View: Spec_______________ Quantity__________

Front View: Spec_______________ Quantity__________

Back View: Spec_______________ Quantity__________

Back View: Spec_______________ Quantity__________

67. ***Refer to the Gresen Spec sheet for the following question***

What are the depths of the Trepans listed above?

Print Reading Skills

68. Refer to Gresen spec sheet number 2.003.070 - 2.003.077 for Trepan Grooves.

What are the minimum and maximum tolerances for diameter Ø A?

For	Min	Max
.071		

Print Reading Skills

69. Refer to Gresen spec sheet number 2.003.070 - 2.003.077 for Trepan Grooves.

What are the minimum and maximum tolerances for diameter Ø B?

For	Min	Max
.075		

Print Reading Skills

70. What is the maximum and minimum size of the tooling holes shown on the bottom view?

____

71. How many tooling holes are shown on the bottom view?

____

72. What is the maximum and minimum depth of the tooling holes shown on the bottom view?

____

73. What is the distance between the tooling holes shown on the bottom view from surface **–E–**?

____

74. The two tooling holes are dimensioned from surface **–H–**. What are the dimensions?

____

____

75. ***Look at the back view.*** What is the vertical dimension from the lower counterbore hole to the centerline of the top Trepan?

____

76. What is the finish required for the relief spec 2.003.049 from surface **–A–**?

Hint: Check the spec sheet.

Section 3: Reasoning Skills

Reasoning Skills

77. *Start-Up Procedures for C.N.C. Machines*

Number the following start up procedure steps in the order that you think is most reasonable. A total of seven steps.

Hint: There are several reasonable ways to number these.

_______ Reset E stop

_______ Assign program

_______ Check oil, grease and coolant

_______ Turn main power on

_______ Check pallet assignment for job

_______ Home out or zero out axes

_______ Verify tool and fixture offsets

Reasoning Skills

78. *End of Shift Shut Down Procedures for C.N.C. Machines*

Number the following end-of-shift shut down procedure steps in the order that you think is most reasonable. A total of four steps.

______ Depress E stop

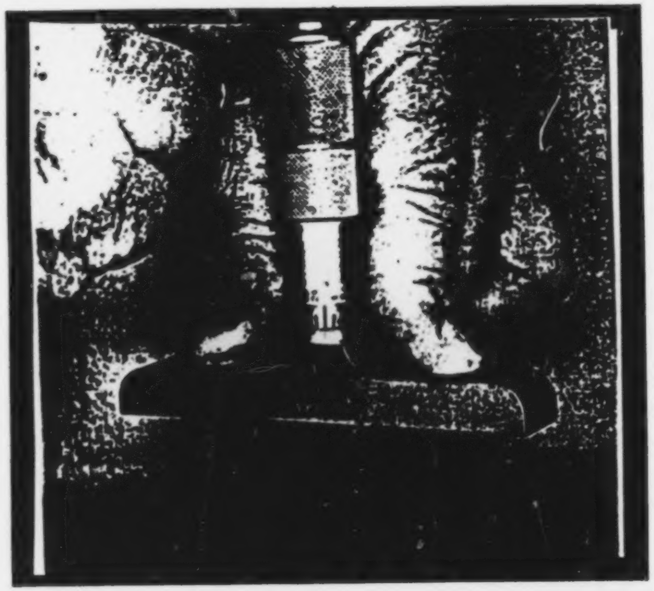
______ Switch main power off (if required)

______ Jog X, Y, Z & B axes off home positions

______ Depress master stop

Identify the Measuring Instruments

Circle the letter that corresponds with the correct instrument term.

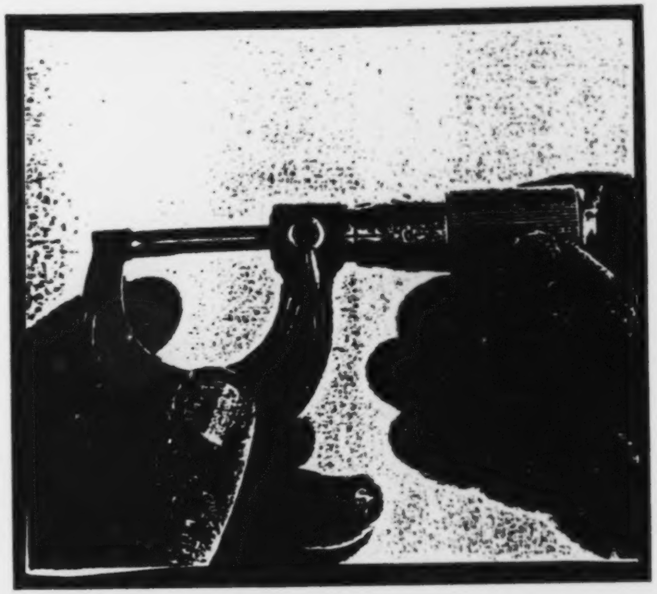


79.

a) Depth micrometer

b) Protractor

c) Height gage



80.

a) Blade micrometer

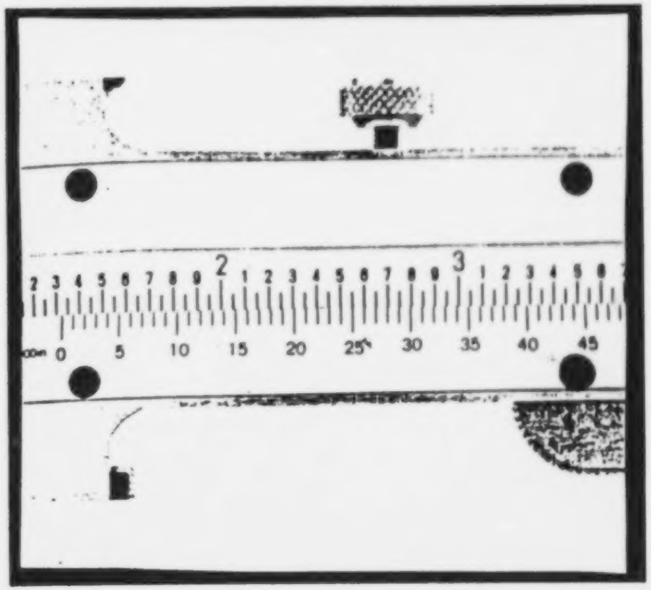
b) Thickness gage

c) One inch micrometer

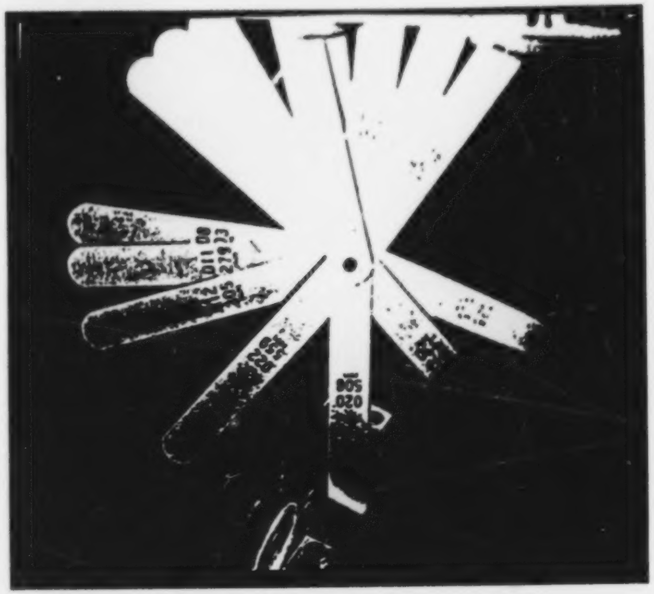
BEST COPY AVAILABLE

Identify the Measuring Instruments

Circle the letter that corresponds with the correct instrument term.


2
3
0
5
10
15
20
25
30
35
40
45

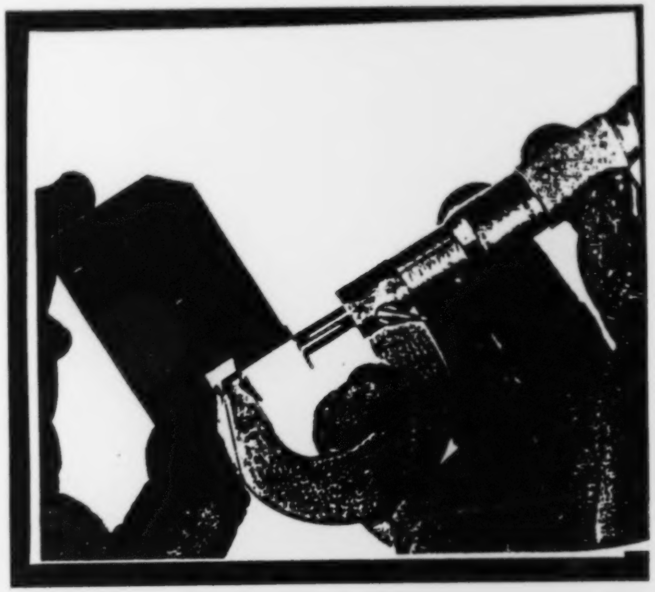
81.
- a) Protractor
- b) Vernier
- c) Scale



82.
- a) Feeler or thickness gage
- b) Dial indicator
- c) Thread gage

Identify the Measuring Instruments

Circle the letter that corresponds with the correct instrument term.

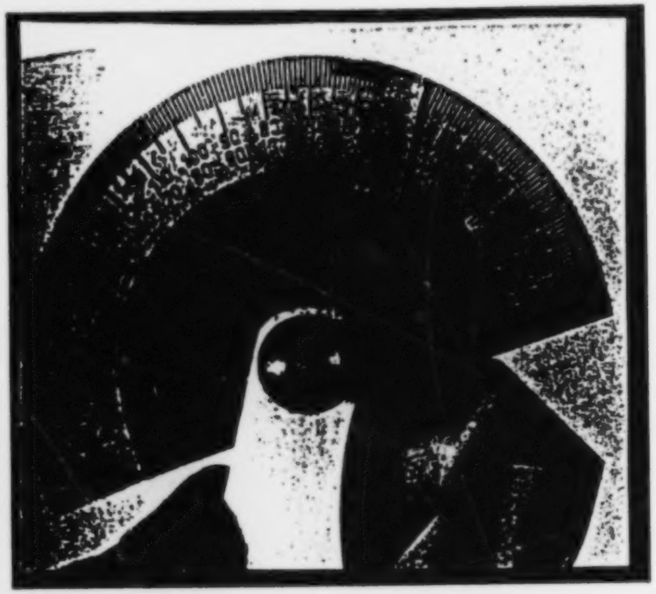


83.

a) Disc micrometer

b) One inch micrometer

c) Blade micrometer



84.

a) Depth micrometer

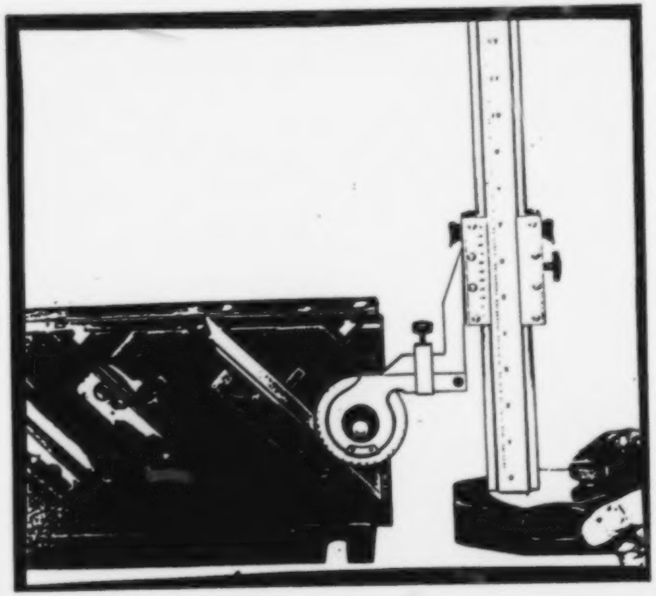
b) Vernier

c) Protractor

BEST COPY AVAILABLE

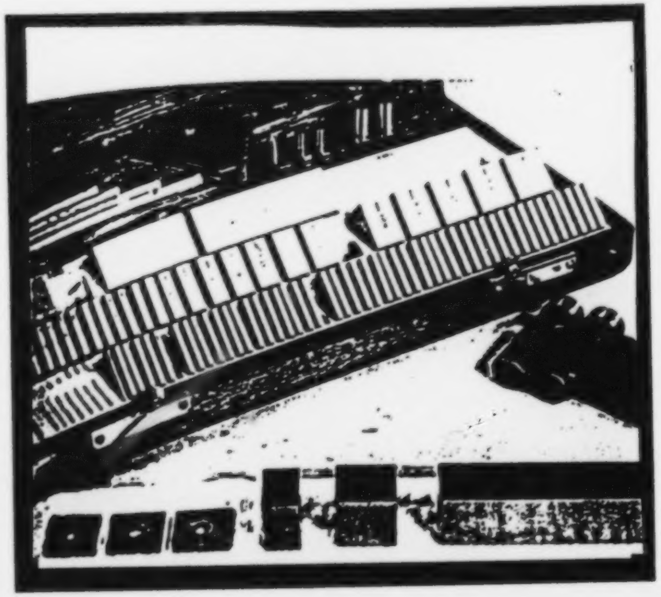
Identify the Measuring Instruments

Circle the letter that corresponds with the correct instrument term.



85.

a) Height gage
b) Thread gage
c) Depth micrometer

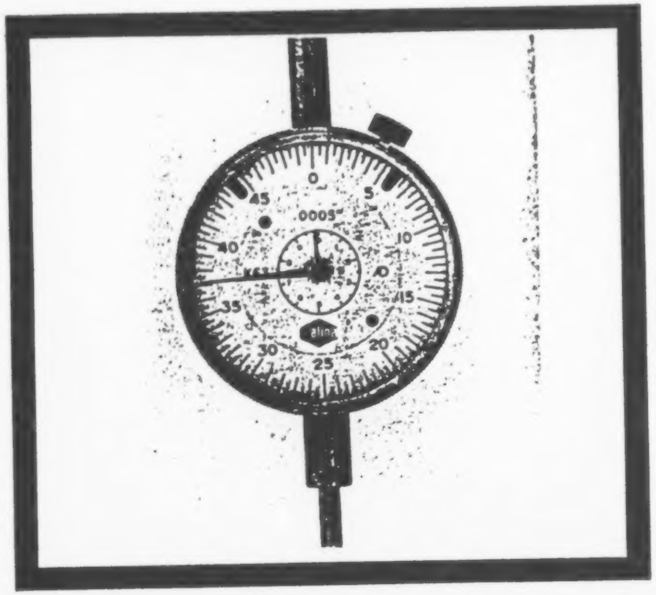


86.

a) Disc micrometer
b) Dial indicator
c) Gage blocks

Identify the Measuring Instruments

Circle the letter that corresponds with the correct instrument term.

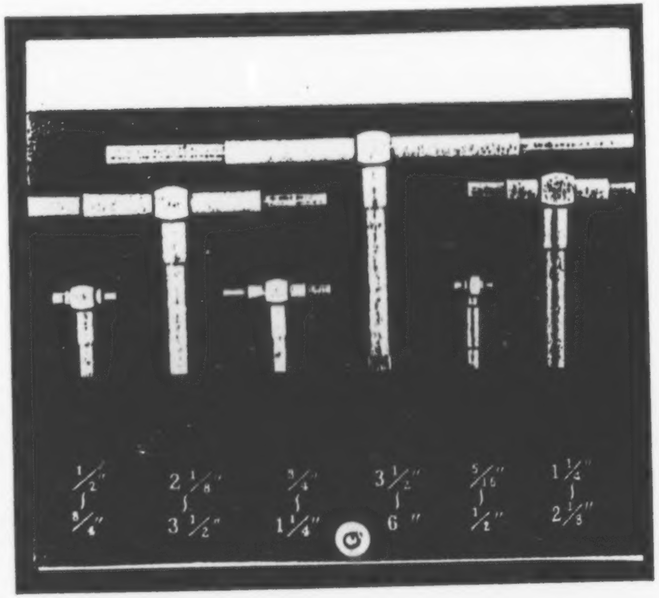


87.

a) Vernier

b) Dial indicator

c) Thread gage


1/2" – 3/4"
2 1/8" – 3 1/2"
3/4" – 1 1/4"
3 1/2" – 6"
5/16" – 1/2"
1 1/4" – 2 1/8"

88.

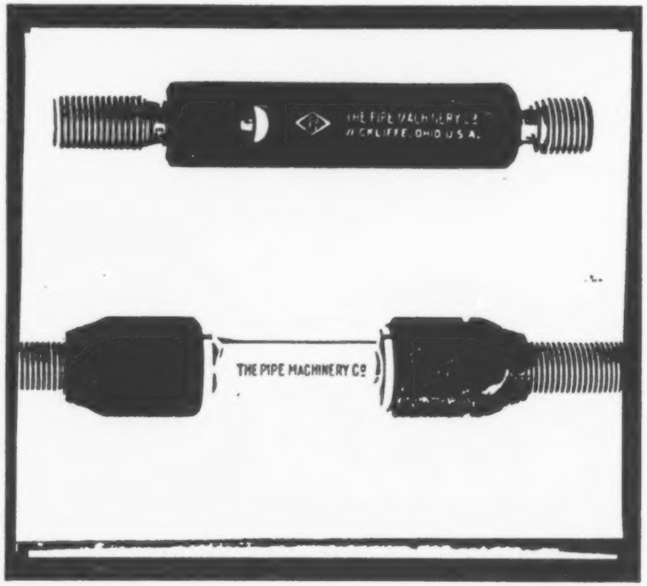
a) Telescopic gages

b) Depth micrometers

c) Protractors

Identify the Measuring Instruments

Circle the letter that corresponds with the correct instrument term.


THE PIPE MACHINERY CO
WICKLIFFE, OHIO U.S.A.
THE PIPE MACHINERY CO

89.

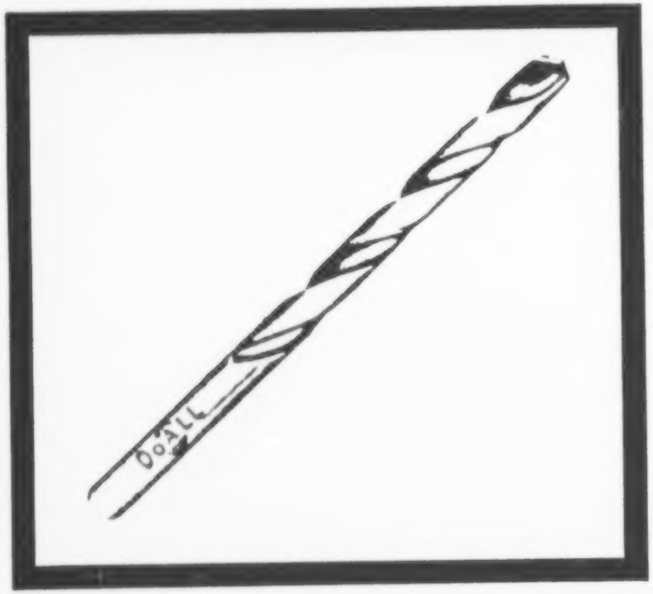
a) Height gages

b) Gage blocks

c) Thread gages

Identify the Tools

Circle the letter that corresponds with the correct tool term.

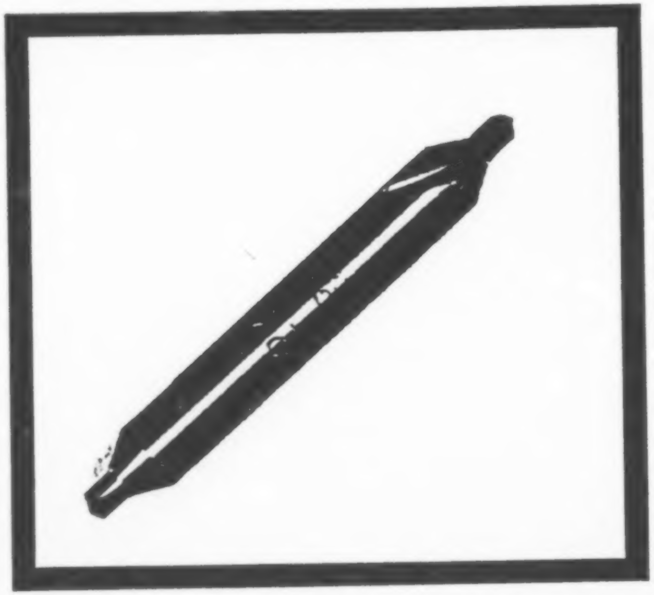


90.

a) Drill

b) Pipe tap

c) Reamer



91.

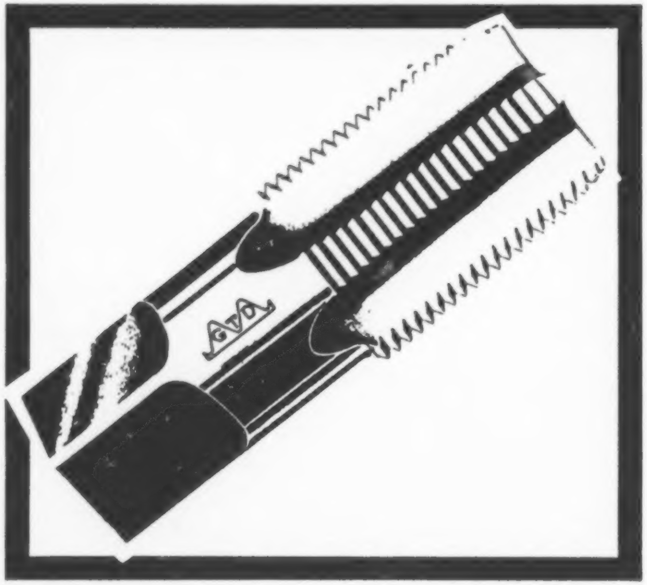
a) Straight thread tap

b) Pipe tap

c) Drill/counterbore combo.

Identify the Tools

Circle the letter that corresponds with the correct tool term.

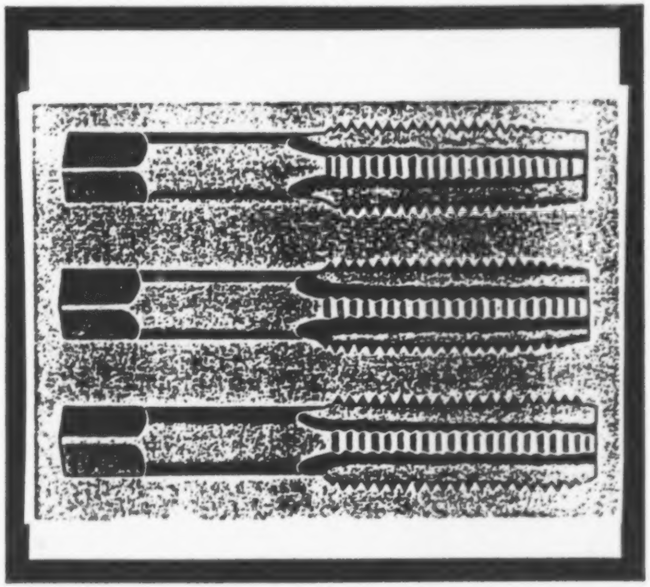


92.

a) Pipe tap

b) Straight shank reamer

c) Drill



93.

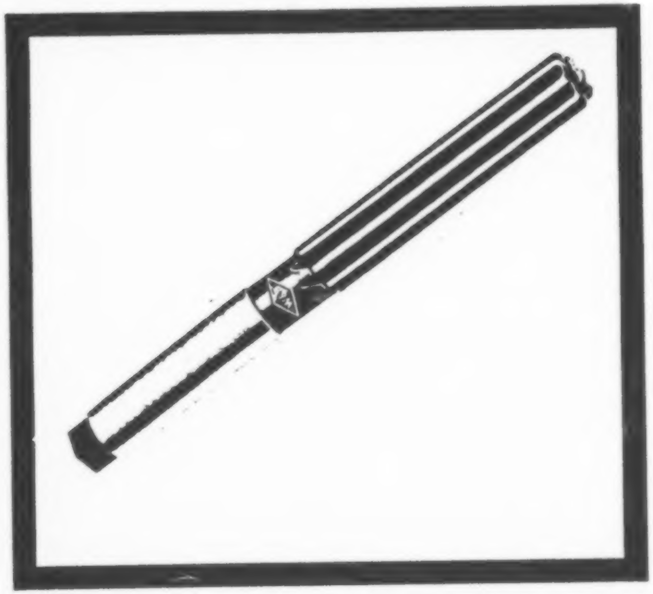
a) Reamer

b) Drill

c) Straight thread tap

Identify the Tools

Circle the letter that corresponds with the correct tool term.



94.

a) Straight thread tap

b) Straight shank reamer

c) Drill

Reasoning Skills

95. Fill in the blanks with your answer.

You bore and ream a hole and find the hole is tapered.

What may have caused the problem?

Reasoning Skills

96. Fill in the blanks with your answer.

You're running a job and notice a feature has chatter.

What would you consider before you change the feeds and speeds?

Reasoning Skills

97. Fill in the blanks with your answer.

You're at a machine and you hear a noise that could lead to tool failure.

How should you react?

____

____

____

____

Basic CNC Operation

Hands-On Assessment

Assessment Tasks 1 - 3

Scoring Key: Your goal is to score a 4 or 5 in each area.
5 = Outstanding 4 = Above Average
3 = Average 2 = Below Average 1 = Poor

The operator...

1. Start up and shut down

A.	Demonstrates start up procedure	1	2	3	4	5
B.	Demonstrates shut down procedure	1	2	3	4	5

2. Routine maintenance

A.	Determines oil levels	1	2	3	4	5
B.	Determines coolant levels	1	2	3	4	5

3. Controls

A.	Identifies manual data input	1	2	3	4	5
B.	Identifies auto run	1	2	3	4	5
C.	Identifies set up modes	1	2	3	4	5
D.	Identifies dry run feature	1	2	3	4	5
E.	Identifies single block feature	1	2	3	4	5

Assessment Tasks 4 - 5

4. Basic editing

A.	Calls up program from memory	1	2	3	4	5
B.	Identifies differences between sub programs and main programs	1	2	3	4	5
C.	Edits main or sub program	1	2	3	4	5
D.	Inserts tool radius and length offsets	1	2	3	4	5
E.	Inserts work coordinate offsets	1	2	3	4	5
F.	Identifies custom macro blocks of information	1	2	3	4	5

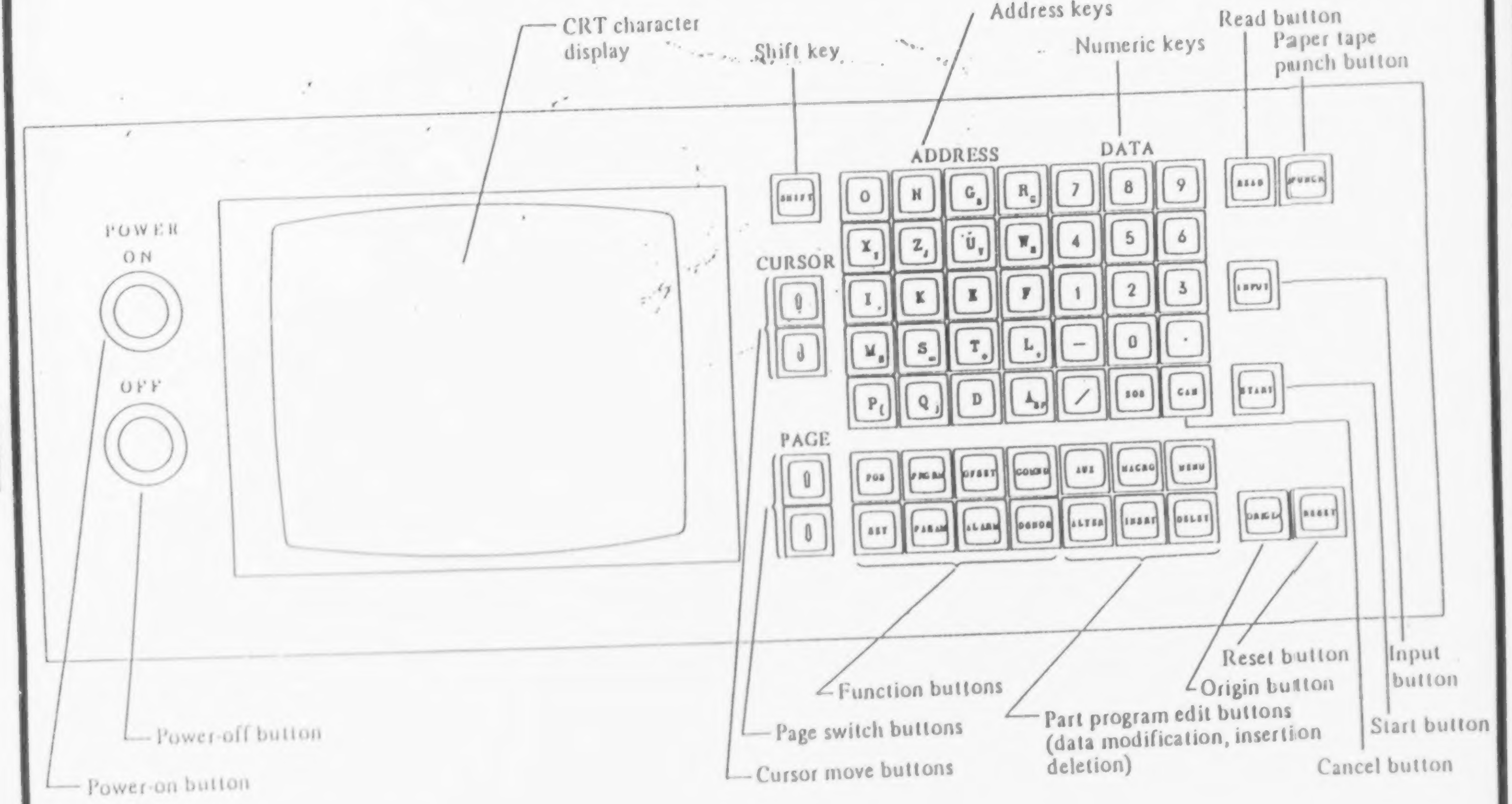
5. Machine set-up

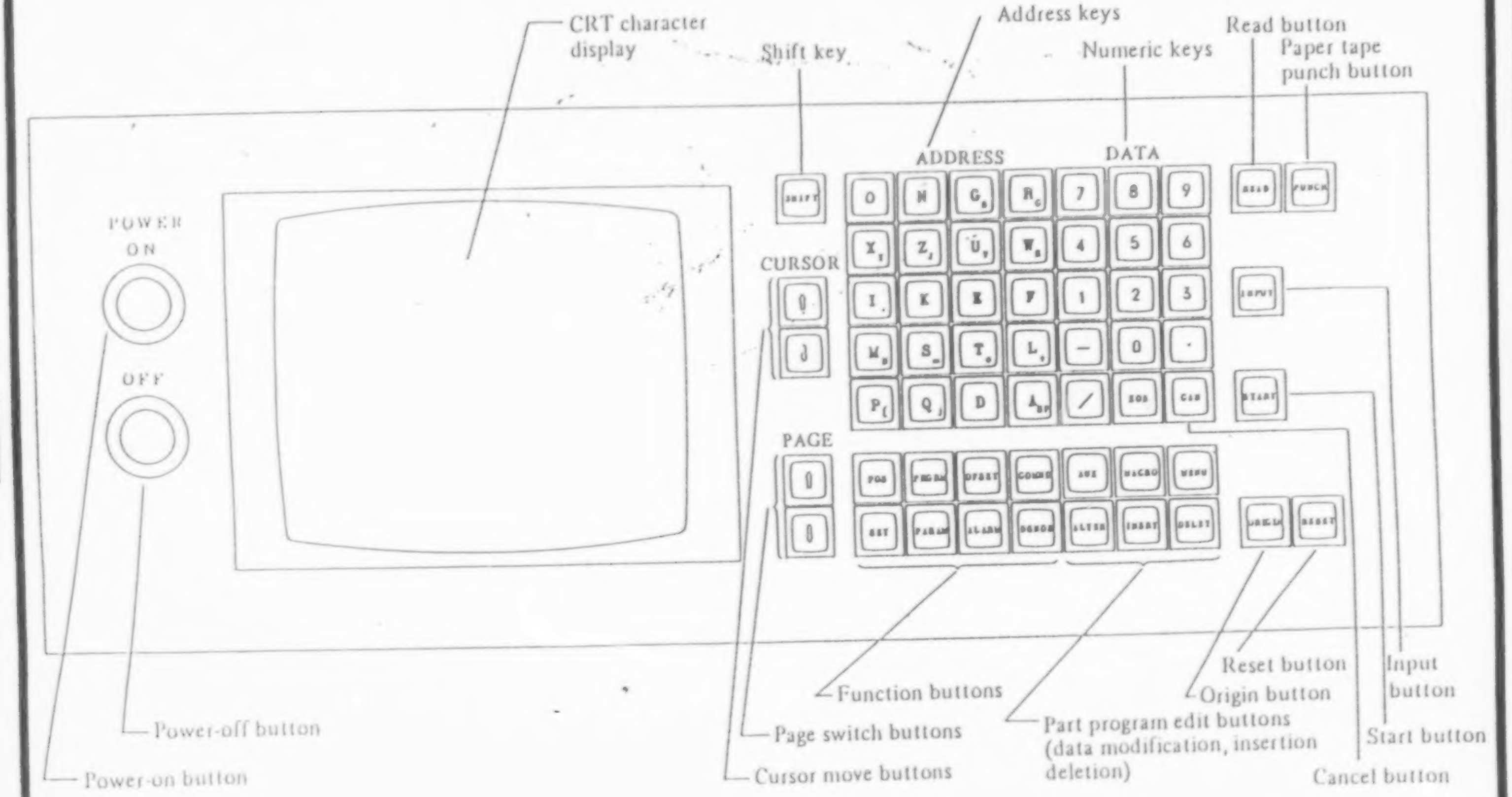
A.	Mill: Sets X-0 , Y-0 and Z-0 Using edge finder or dial indicator or probe Lathe: Sets X-0 and Z-0 Using probe or tools	1	2	3	4	5
B.	Pre-sets tools with tool presetter [optional: use touch off method]	1	2	3	4	5
C.	Utilizes one or more work coordinate offset G54 – G59	1	2	3	4	5
D.	Identifies tool holder styles and numbers Mill: BT, CAT, Milacron, etc. Lathe: right or left hand tool holder	1	2	3	4	5
E.	Explains proper part loading and holding procedures	1	2	3	4	5

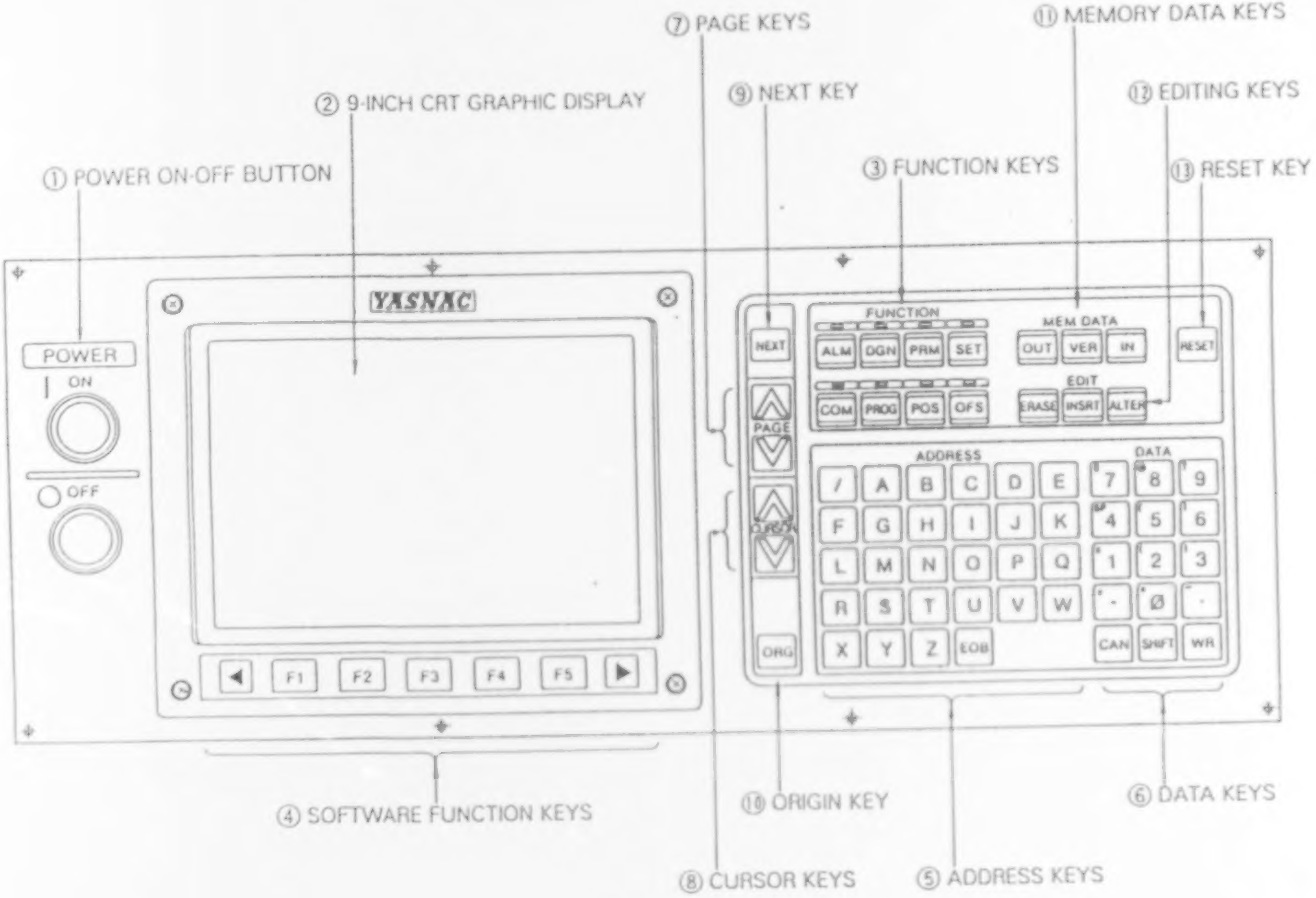
Controls Descriptions

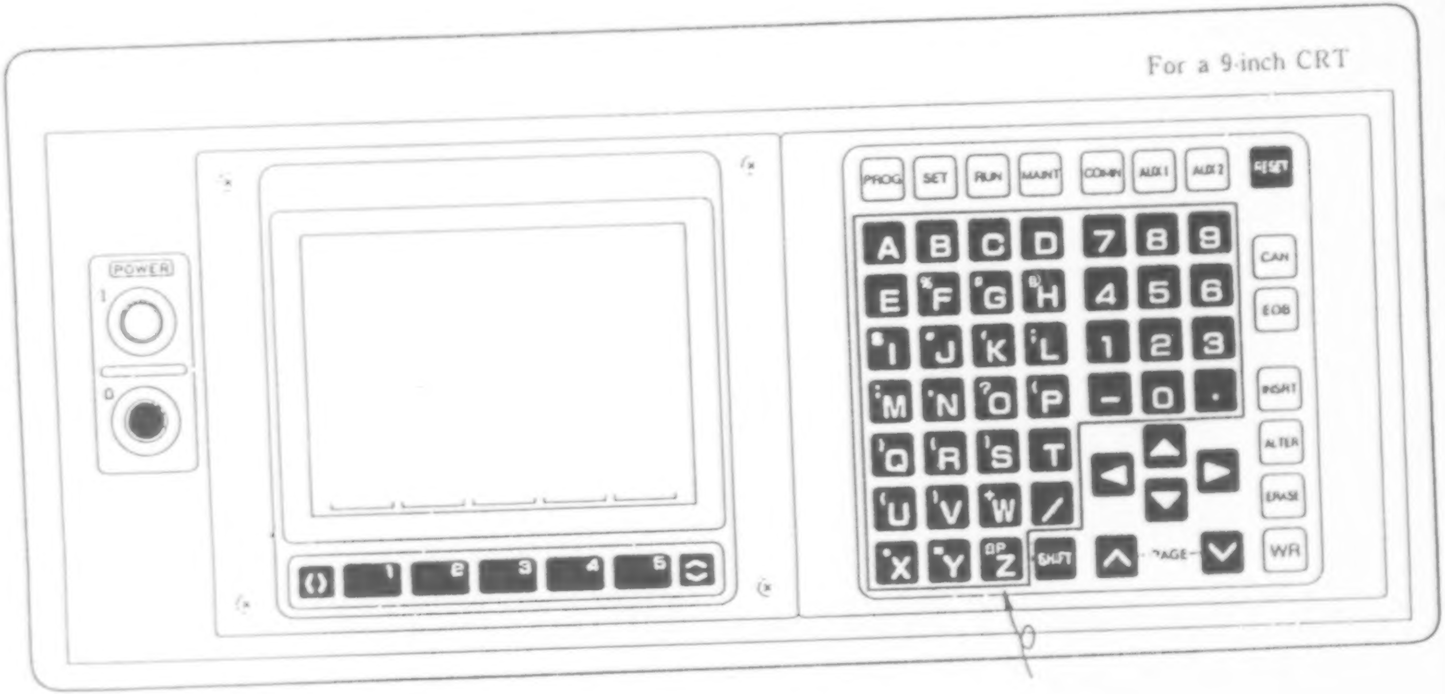
The following descriptions depict similar controls found in your work area. Use the descriptions to match the illustrations found on pages 1 through 14.

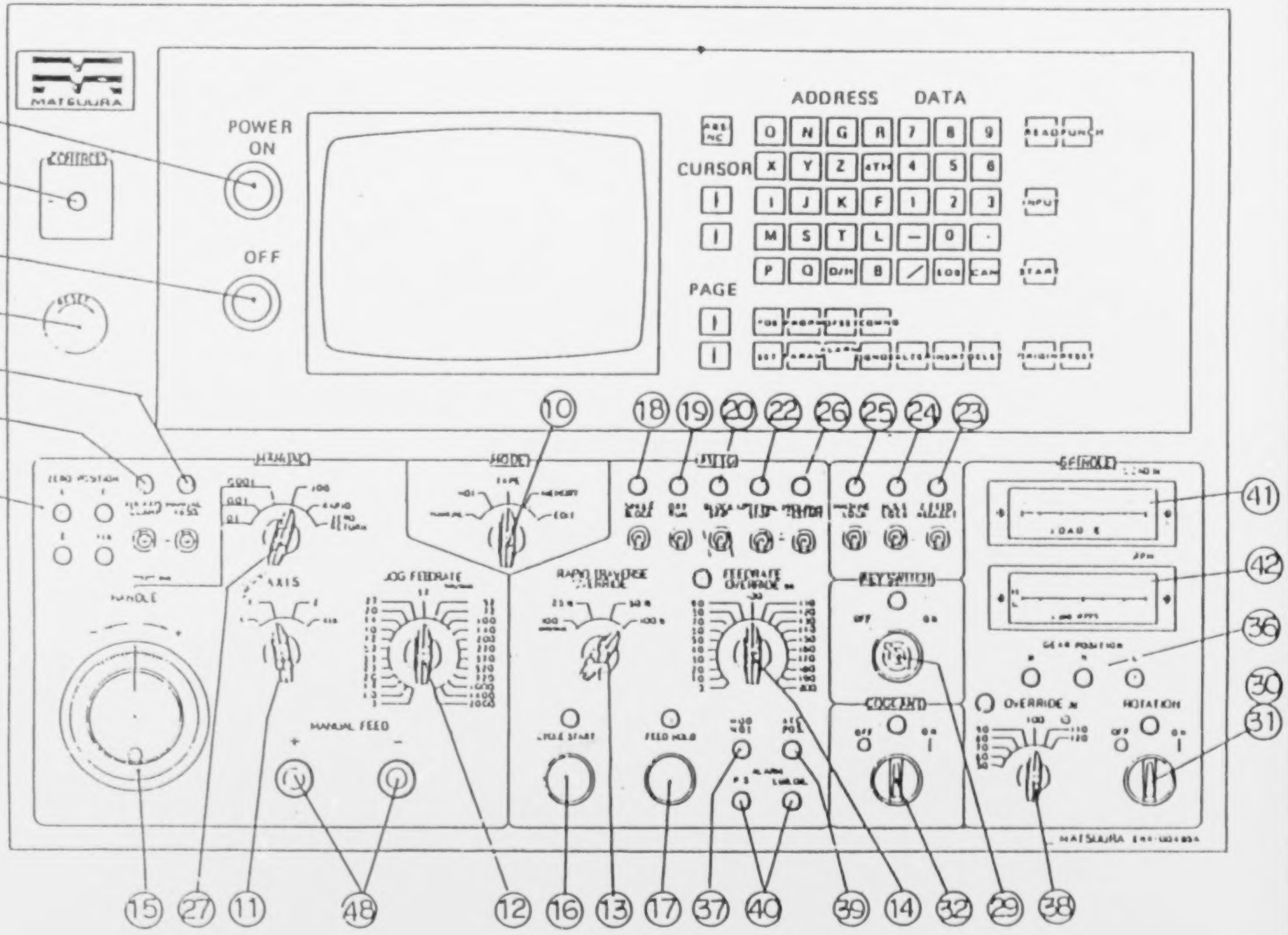
1. Nakamuura, Fanuc, TM System 6
2. Matsuura, Fanuc, System 6
3. Methods, Lathe/Slant 50, Yasnac
4. Matsuura, RATTF, Yasnac, with pallet shuttle
5. Matsuura, MC600 VF, without shuttle, w/4th Axes
6. K.T. 200 KT D Control
    - KT D Control APL
    - KT B Control
7. Mori Seiki, Partner M300, Fanuc MFM4
8. Niigata, Fanuc 15M, HCNiigata PN40
9. Niigata, Fanuc 11M V20, with mounting system
10. Niigata, Fanuc V50 - V70, 6 Pallet
11. Allen-Bradley, Hardinge, Relief Cell
12. Cincinnati, Mylacron, Gear Lathe
13. Parlec Tool Presetter
14. Cincinnati, Acramatic


CRT character display
Shift key
Address keys
Numeric keys
Read button
Paper tape punch button
ADDRESS
DATA
POWER
ON
OFF
CURSOR
PAGE
SHIFT
O
N
G
R
7
8
9
X
Z
U
W
4
5
6
I
K
E
F
1
2
3
M
S
T
L
—
0
.
P
Q
D
A
/
EOB
CAN
READ
PUNCH
INPUT
START
POS
PRGRM
OFSET
COMND
AUX
MACRO
MENU
SET
PARAM
ALARM
DGNOS
ALTER
INSRT
DELET
ORGIN
RESET
Reset button
Input button
Origin button
Function buttons
Part program edit buttons (data modification, insertion deletion)
Start button
Page switch buttons
Cancel button
Cursor move buttons
Power-off button
Power-on button


CRT character display
Shift key
Address keys
Numeric keys
Read button
Paper tape punch button
ADDRESS
DATA
POWER ON
OFF
CURSOR
PAGE
Reset button
Origin button
Input button
Function buttons
Start button
Part program edit buttons (data modification, insertion deletion)
Power-off button
Page switch buttons
Cursor move buttons
Cancel button
Power-on button

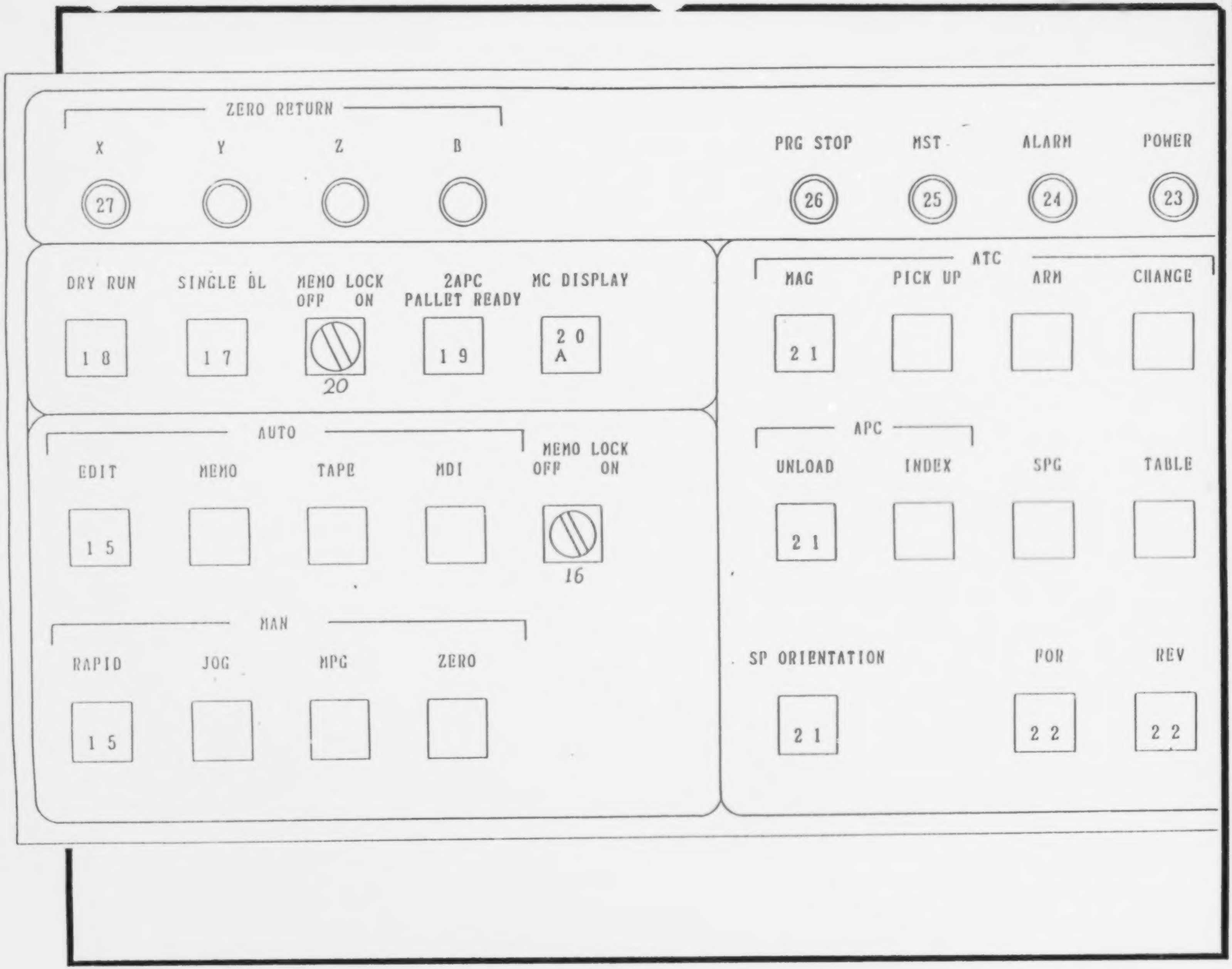

⑦ PAGE KEYS
⑪ MEMORY DATA KEYS
② 9-INCH CRT GRAPHIC DISPLAY
⑨ NEXT KEY
⑫ EDITING KEYS
① POWER ON-OFF BUTTON
③ FUNCTION KEYS
⑬ RESET KEY
YASNAC
POWER
ON
OFF
F1
F2
F3
F4
F5
NEXT
PAGE
CURSOR
ORG
FUNCTION
ALM
DGN
PRM
SET
COM
PROG
POS
OFS
MEM DATA
OUT
VER
IN
EDIT
ERASE
INSRT
ALTER
RESET
ADDRESS
/
A
B
C
D
E
F
G
H
I
J
K
L
M
N
O
P
Q
R
S
T
U
V
W
X
Y
Z
EOB
DATA
7
8
9
4
5
6
1
2
3
-
0
.
CAN
SHIFT
WR
④ SOFTWARE FUNCTION KEYS
⑩ ORIGIN KEY
⑥ DATA KEYS
⑧ CURSOR KEYS
⑤ ADDRESS KEYS

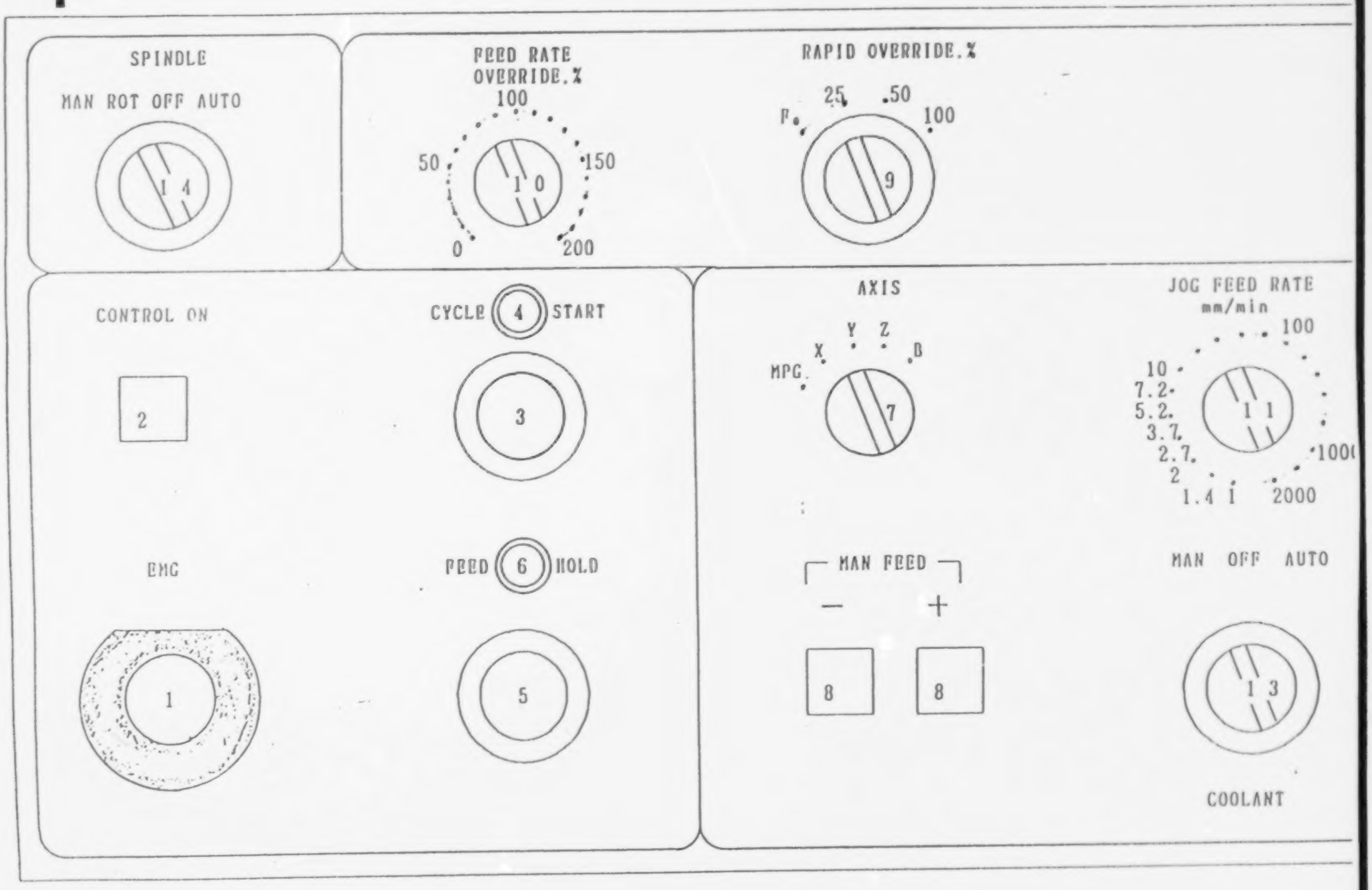

For a 9-inch CRT
POWER
PROG
SET
RUN
MAINT
COMM
AUX 1
AUX 2
RESET
A
B
C
D
7
8
9
E
F
G
H
4
5
6
I
J
K
L
1
2
3
M
N
O
P
-
0
.
Q
R
S
T
U
V
W
/
X
Y
Z
SHIFT
PAGE
CAN
EOB
INSRT
ALTER
ERASE
WR
1
2
3
4
5


MATSUURA
POWER ON
OFF
CONTROL
RESET
ADDRESS
DATA
CURSOR
PAGE
O N G R 7 8 9
X Y Z 4TH 4 5 6
I J K F 1 2 3
M S T L — 0 .
P Q D/H B / EOB CAN
READ PUNCH
INPUT
START
ORIGIN RESET
HANDLE
AXIS
JOG FEEDRATE
MANUAL FEED
+
−
MODE
TAPE
MEMORY
EDIT
RAPID TRAVERSE OVERRIDE
FEEDRATE OVERRIDE %
CYCLE START
FEED HOLD
ALARM
KEY SWITCH
OFF
ON
COOLANT
SPINDLE
LOAD
GEAR POSITION
OVERRIDE %
ROTATION
MATSUURA
6
8
7
9
21
28
34
10
18
19
20
22
26
25
24
23
41
42
36
30
31
15
27
11
48
12
16
13
17
37
40
39
14
32
29
38

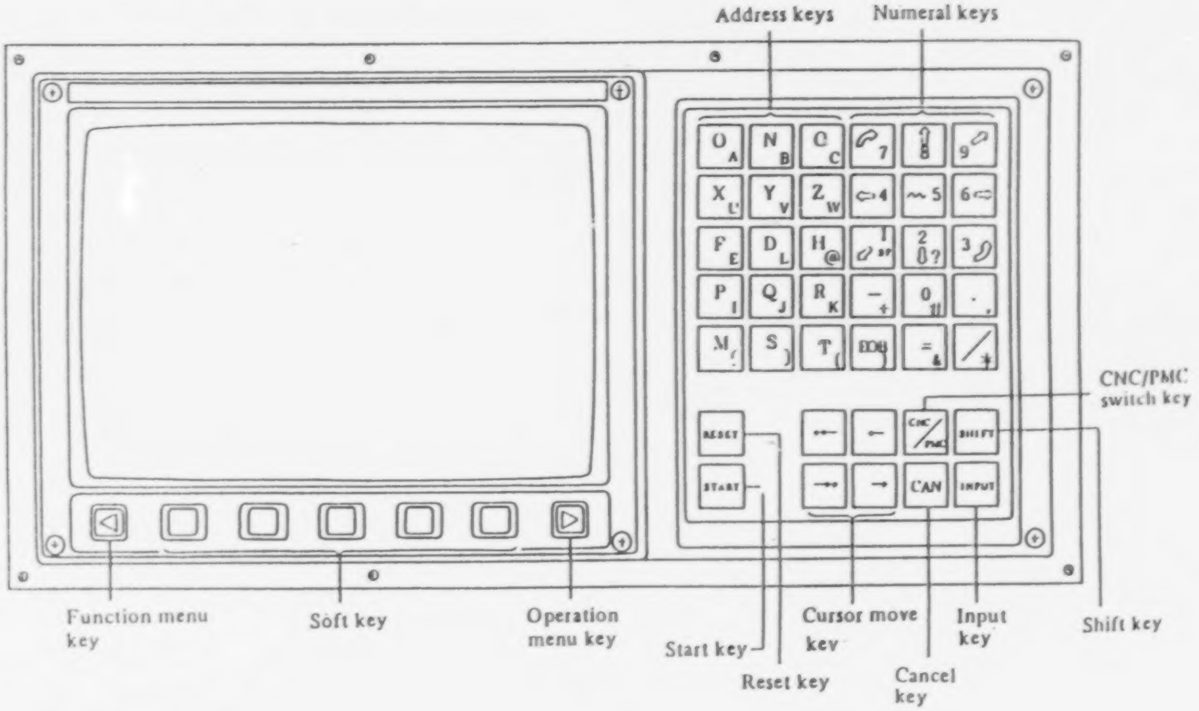
Matsuura MC600 VF

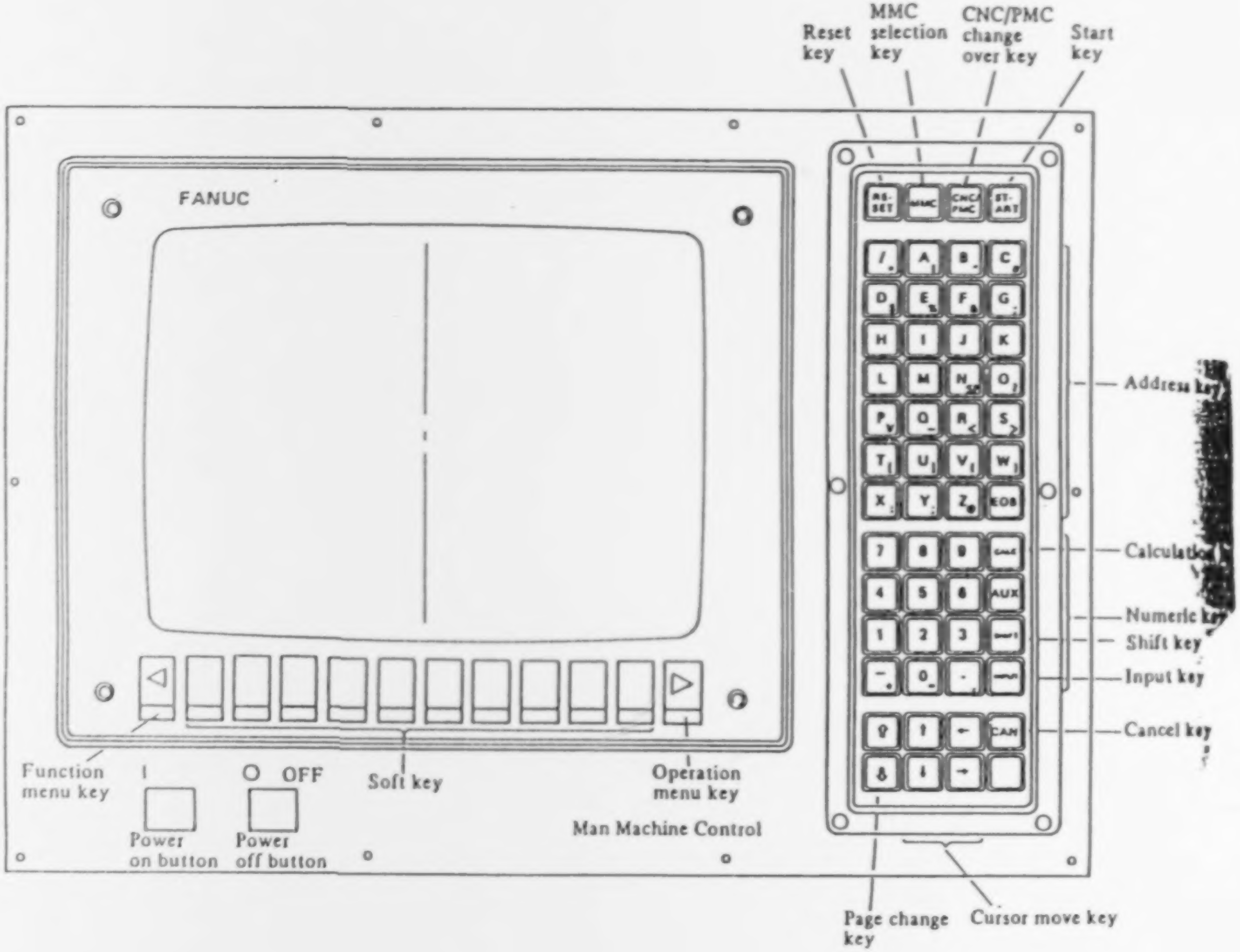
Yasnac


ZERO RETURN
X
Y
Z
B
27
PRG STOP
MST
ALARM
POWER
26
25
24
23
DRY RUN
SINGLE BL
MEMO LOCK
OFF ON
2APC
PALLET READY
MC DISPLAY
18
17
20
19
20
A
ATC
MAG
PICK UP
ARM
CHANGE
21
AUTO
EDIT
MEMO
TAPE
MDI
MEMO LOCK
OFF ON
15
16
APC
UNLOAD
INDEX
SPG
TABLE
21
MAN
RAPID
JOG
MPG
ZERO
15
SP ORIENTATION
FOR
REV
21
22
22


SPINDLE
MAN ROT OFF AUTO
14
FEED RATE OVERRIDE.%
0
50
100
150
200
10
RAPID OVERRIDE.%
F
25
50
100
9
CONTROL ON
2
CYCLE
4
START
3
EMG
1
FEED
6
HOLD
5
AXIS
MPG
X
Y
Z
B
7
JOG FEED RATE mm/min
1
1.4
2
2.7
3.7
5.2
7.2
10
100
1000
2000
11
MAN FEED
−
+
8
8
MAN OFF AUTO
13
COOLANT

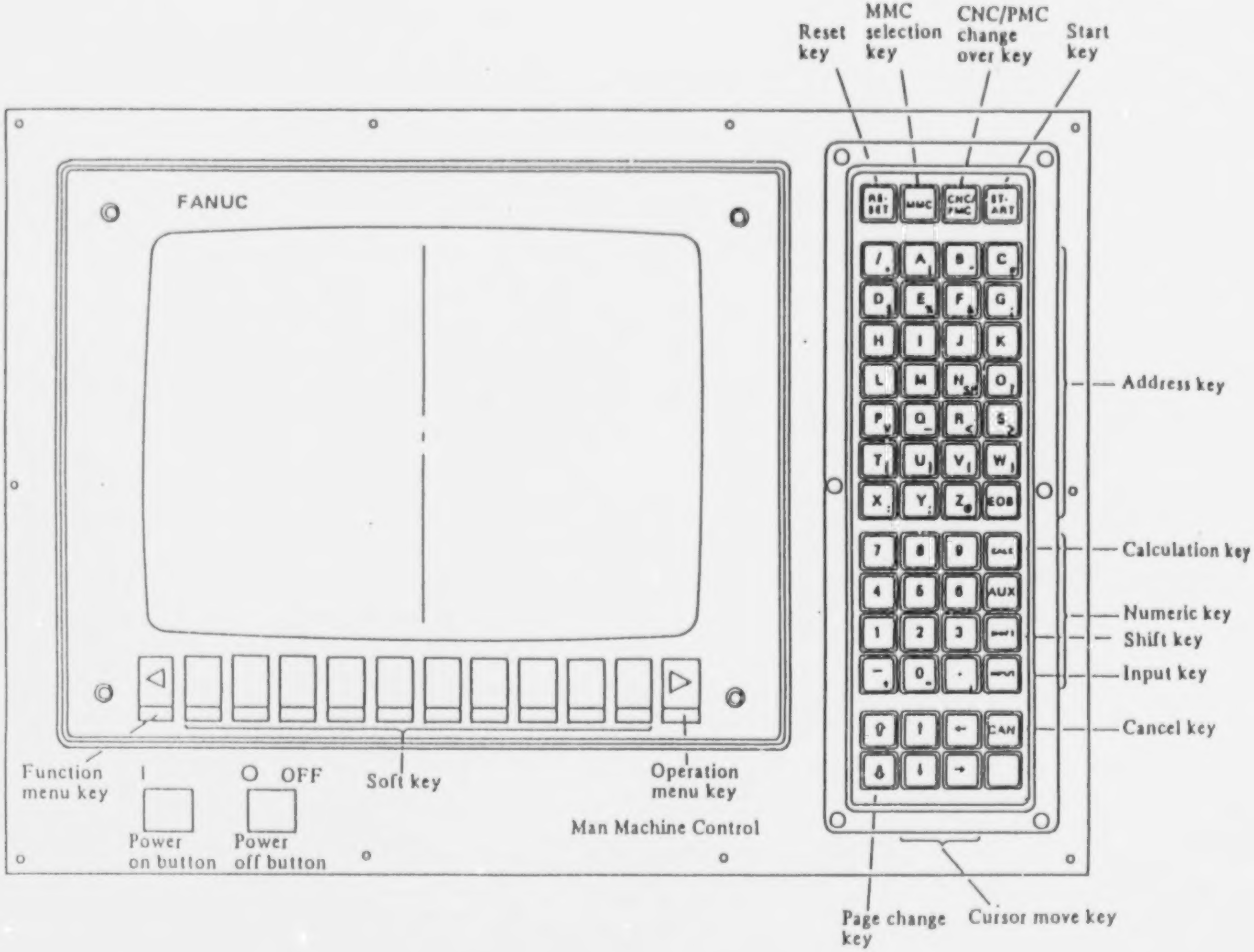
K.T. 200


Address keys
Numeral keys
CNC/PMC switch key
Function menu key
Soft key
Operation menu key
Start key
Reset key
Cursor move key
Cancel key
Input key
Shift key
RESET
START
CAN
INPUT
SHIFT

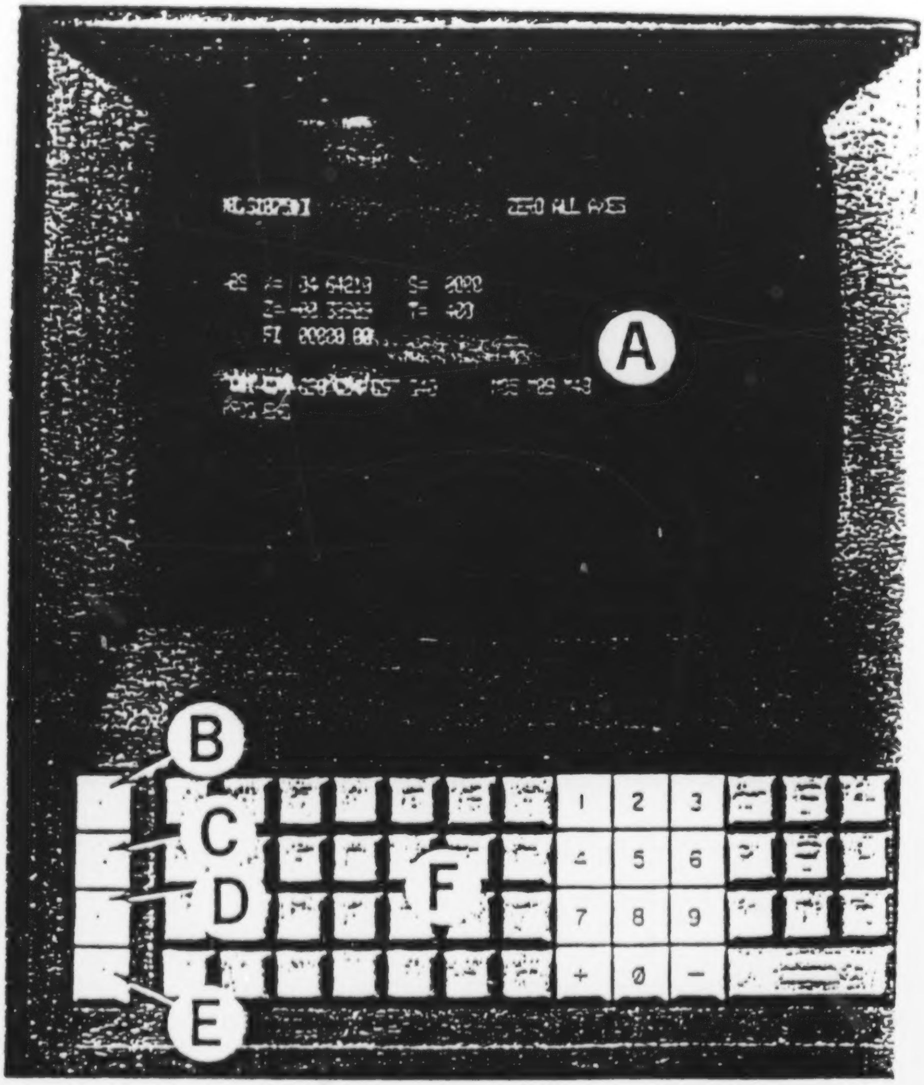

Reset key
MMC selection key
CNC/PMC change over key
Start key
FANUC
RE-SET
MMC
CNC/PMC
ST-ART
Address key
Calculation
Numeric key
Shift key
Input key
Cancel key
AUX
EOB
CAN
Function menu key
I
O OFF
Soft key
Operation menu key
Power on button
Power off button
Man Machine Control
Page change key
Cursor move key

Niigata - Fanuc 11M

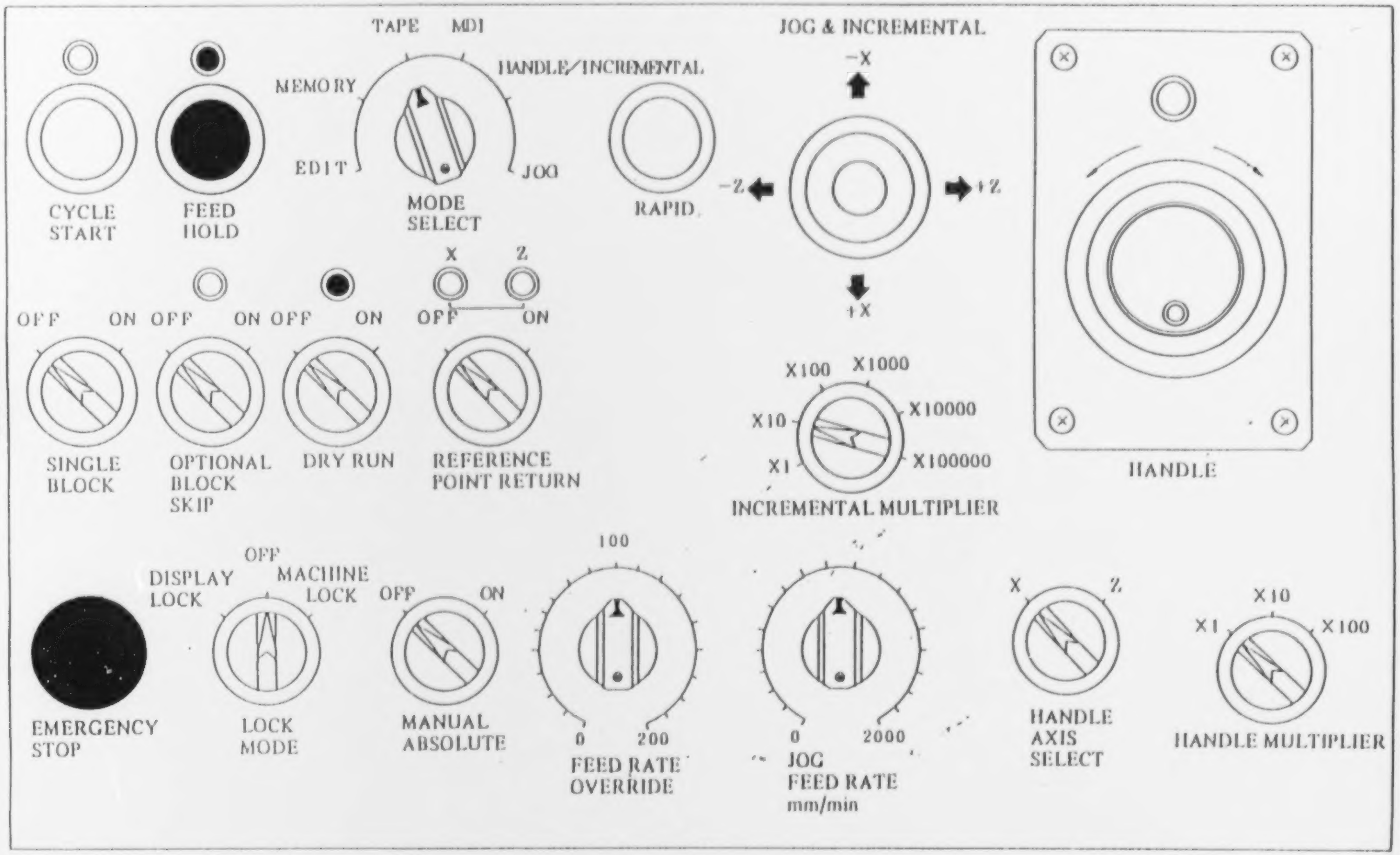
Not shown in available illustrations.

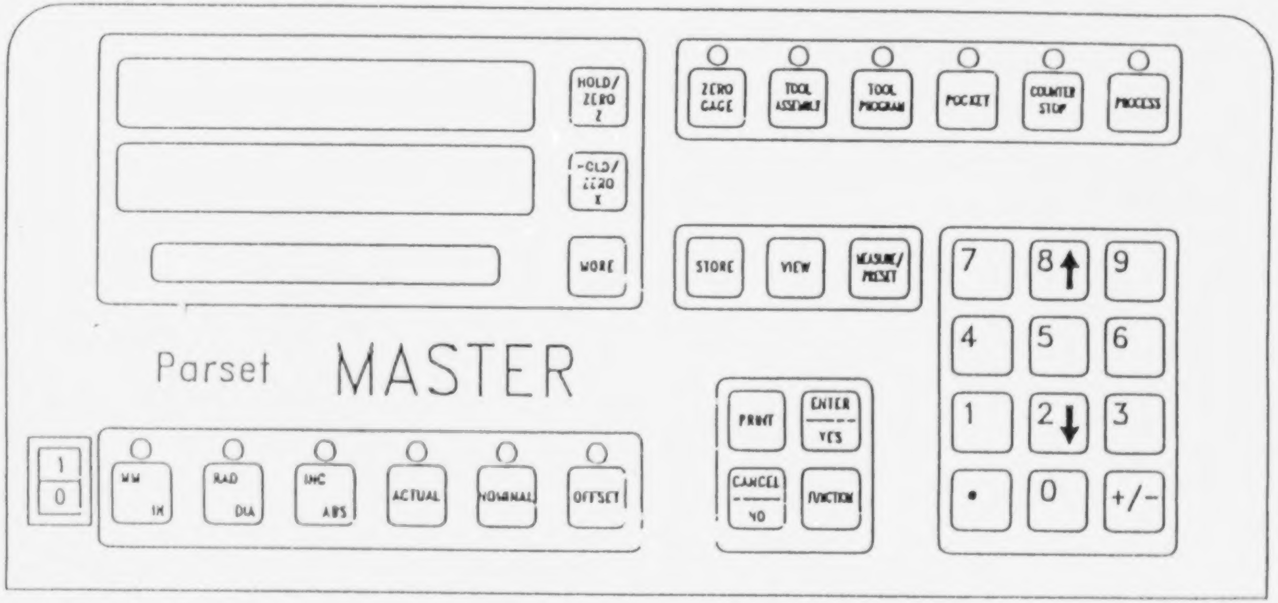

Reset key
MMC selection key
CNC/PMC change over key
Start key
FANUC
RE-SET
MMC
CNC/PMC
ST-ART
Address key
Calculation key
Numeric key
Shift key
Input key
Cancel key
CALC
AUX
CAN
EOB
Function menu key
I
O OFF
Soft key
Operation menu key
Power on button
Power off button
Man Machine Control
Page change key
Cursor move key

Allen Bradley - Hardinge

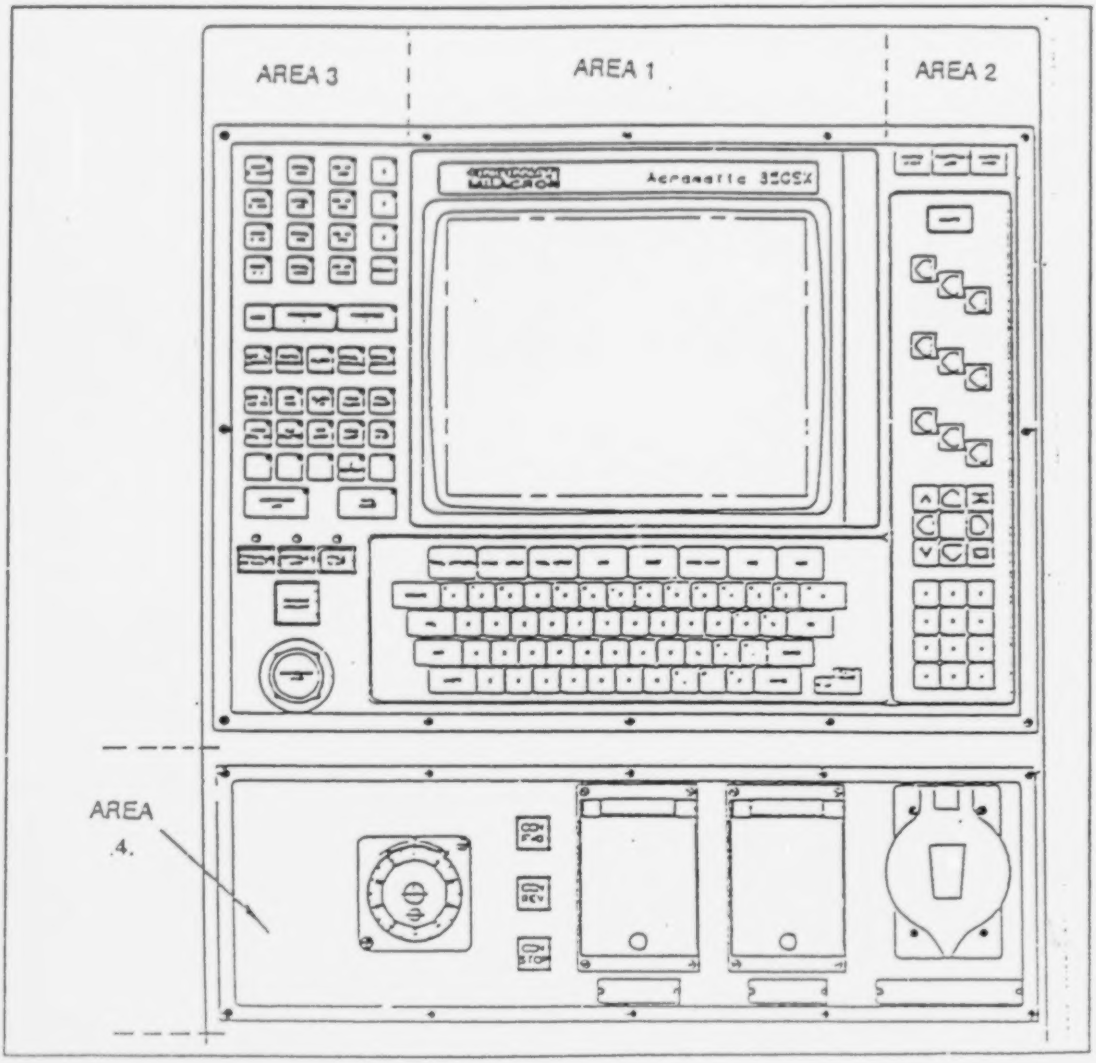

ZERO ALL AXES
A
B
C
D
E
F
1
2
3
4
5
6
7
8
9
+
0
−

BEST COPY AVAILABLE


CYCLE START
FEED HOLD
TAPE
MDI
MEMORY
HANDLE/INCREMENTAL
EDIT
JOG
MODE SELECT
RAPID
JOG & INCREMENTAL
−X
−Z
+Z
+X
HANDLE
X
Z
OFF
ON
OFF
ON
OFF
ON
OFF
ON
SINGLE BLOCK
OPTIONAL BLOCK SKIP
DRY RUN
REFERENCE POINT RETURN
X100
X1000
X10
X10000
X1
X100000
INCREMENTAL MULTIPLIER
DISPLAY LOCK
OFF
MACHINE LOCK
OFF
ON
100
0
200
0
2000
X
Z
X10
X1
X100
EMERGENCY STOP
LOCK MODE
MANUAL ABSOLUTE
FEED RATE OVERRIDE
JOG FEED RATE mm/min
HANDLE AXIS SELECT
HANDLE MULTIPLIER


HOLD/ ZERO Z
HOLD/ ZERO X
MORE
ZERO GAGE
TOOL ASSEMBLY
TOOL PROGRAM
POCKET
COUNTER STOP
PROCESS
STORE
VIEW
MEASURE/ PRESET
7
8
9
4
5
6
1
2
3
•
0
+/-
Parset MASTER
PRINT
ENTER YES
CANCEL NO
FUNCTION
1
0
MM IN
RAD DIA
INC ABS
ACTUAL
NOMINAL
OFFSET

Cincinnati Acramatic



AREA 3
AREA 1
AREA 2
Acramatic 350SX
AREA .4.

401